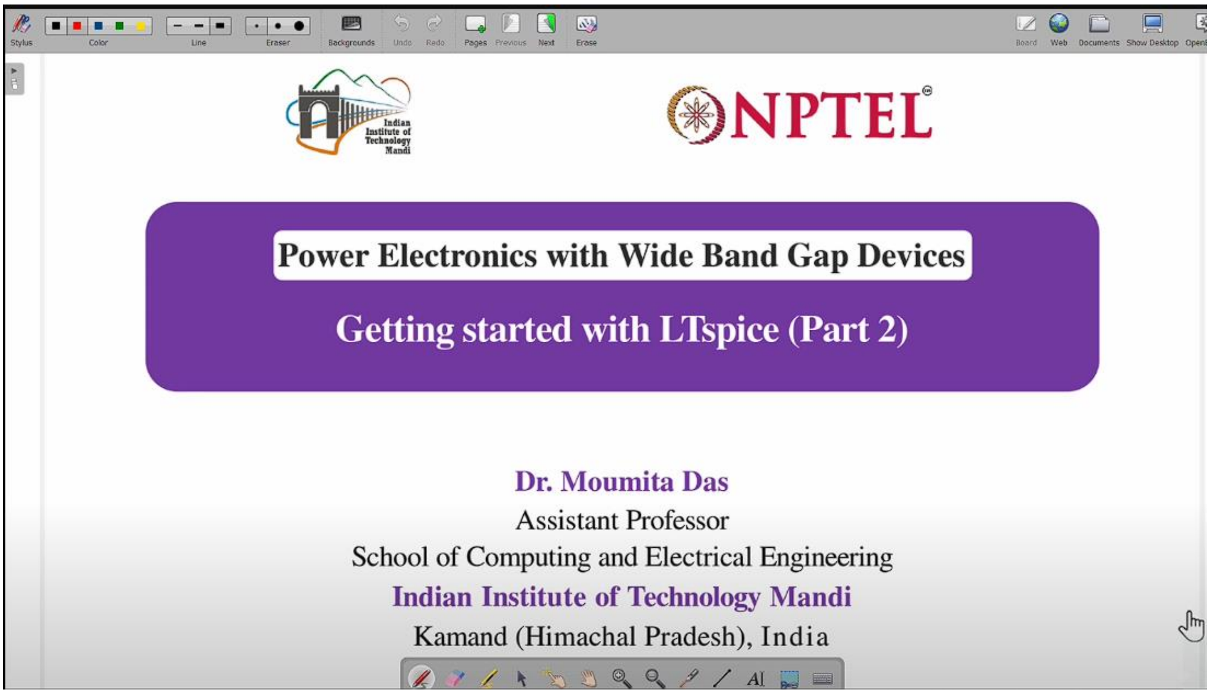
Power Electronics with Wide Bandgap Devices
Dr. Moumita das
School of Computing and Electrical Engineering
Indian Institute of Technology, Mandi

Lecture-27
GETTING STARTED WITH LTSPICE (Part 2)

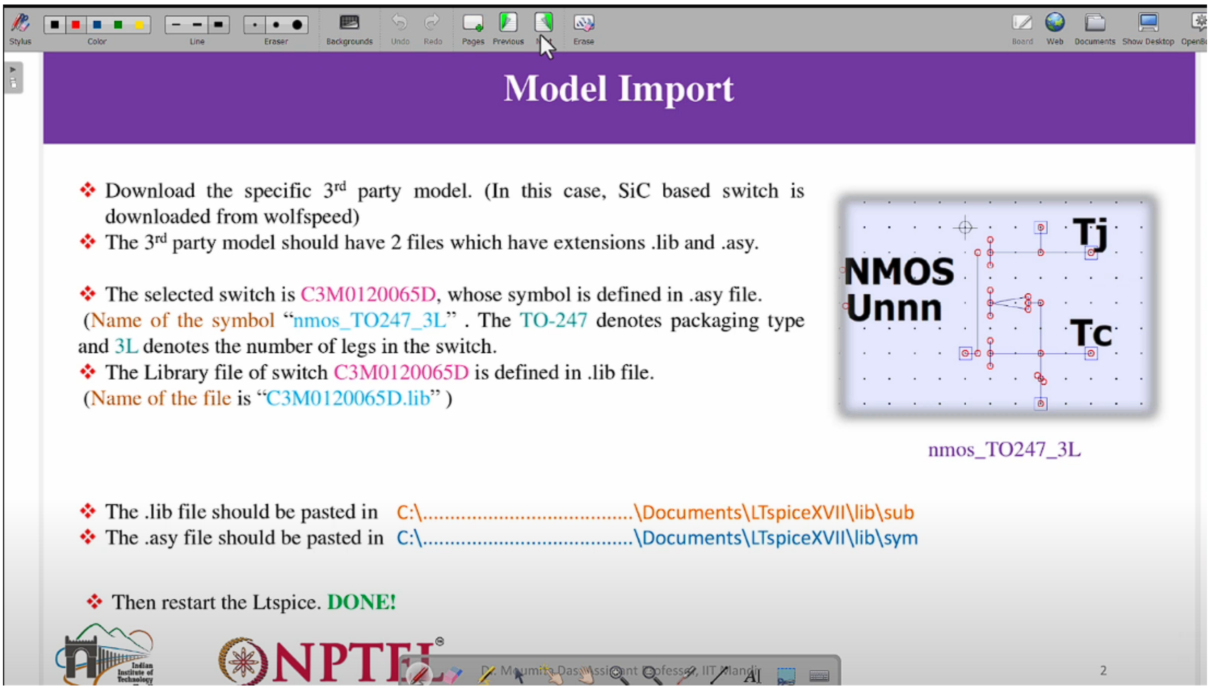
GETTING STARTED WITH LTSPICE (Part 2)

Refer Slide (0:27)



Welcome to the course on power electronics with wide band gap devices. Today I will continue discussion of the LTSpice simulation. So in the last lecture I have already given basic of the LTSpice simulation. Today I am going to show you importing a device means the switch from particular website, it can be GaN switch or silicon carbide since we are discussing about wide band gap devices and use it in simulation to see performance of the converter which will be close to the practical converter operation. That is why we have chose this LTSpice simulation.

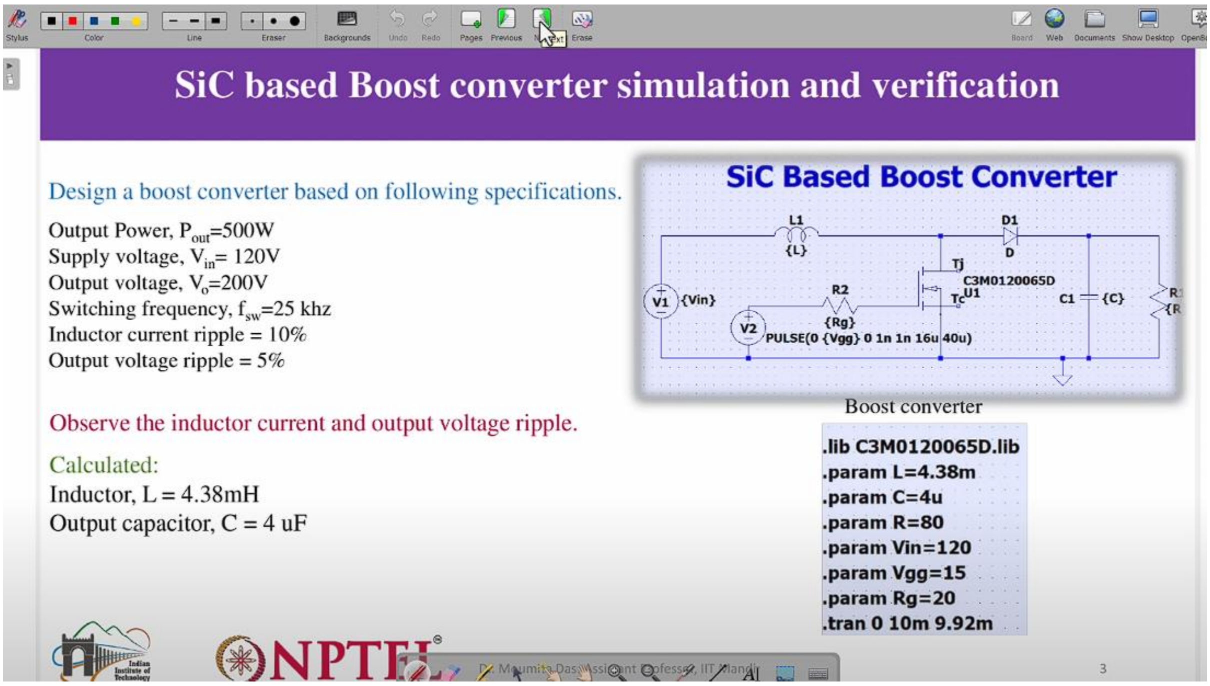
Refer Slide (1:03)



So, now in the last lecture, I have already discussed that how to import a model. Model means model of particular switch. So, if you are choosing silicon carbide choose silicon carbide switch and then if you are choosing GaN then choose particular website. To get the model of that particular device. You can see here, so this already I have discussed, I will just go through this again because this is the thing which I will be needing for today's lecture. So download the specific third-party model. So whatever switch you are assuming that it will be suitable for your converter. And whatever converter you are choosing for your work that also you need to fix first. So this is the one so you can see here this silicon carbide switch is considered in this particular simulation. So now it is downloaded from wolfspeed so you can see here wolfspeed this downloading is given for this silicon carbide switch. So, the third party model should have two files which have extension .lib and .asy as I told you earlier. So, now this is the switch is selected from Wolfspeed. So, C3M0120065D. This symbol is defined in .asy file. So, .asy file is used for this particular switch and then Name of the symbol is given NMOS TO247-3L. The TO247 defines the packaging type. Now packaging can be anything; it can be SMD, through hole anything. So in your application whatever packaging you are using or is suitable that you select. Because what happens you know this packaging will cause difference in the parasitic parameter presence in the device, so whenever you will be selecting any packaging if that is not the actual package which you are going to use in your simulation then that will give the difference in result So, that is why just see whether you are going to consider the SMD kind of packaging to design compact kind of converter or through hole just to know that performance of the converter. So, this is the packaging is selected for this particular device. Now, the library file of this Wolfspeed switch is defined in .lib file, and the name of the file is kept it is same as the name of the device .lib. This is how you should be defining your device. You can choose any other device maybe silicon carbide or maybe you can select GaN device and follow the same procedure to see whether you are able to simulate or not. Okay? now this .lib file should be pasted in this particular address, you can see here in this is we have used C drive, so C drive and then you our LTSpice we have kept in the documents section so under documents the LTSpice we have and they are under library so we have kept this particular device and if it is .lib if it is .asy then you can see that this is under symbol this .asy file is kept. So once you follow this procedure the entire procedure, then what you have to do? You have to restart the LTSpice.

Because you are importing a new device. You have included in the library of the existing LTSpice. So then in order to consider this device for simulation, you need to restart it so that, when you start this LTSpice it will be considered in the library then you can use it as other components in the LTSpice software. Okay?

Refer Slide (5:27)



Now next is that your model is imported so now choose a converter. Here for basic understanding of the simulation, I have kept boost converter so you can see we have actually considered this boost converter so this boost converter it's having following specifications which we are actually using for the simulation. So, you can see here this output power is 500 watts and the input supply voltage is 120 volts. Output voltage of the converter is 200 volts. Switching frequency is considered 25 kilohertz. Now this is something which you can check whether you can you want to simulate it for higher frequency or not because you know wide band gap device means we want to operate the converter at higher frequency. So you can use higher frequency here to see.

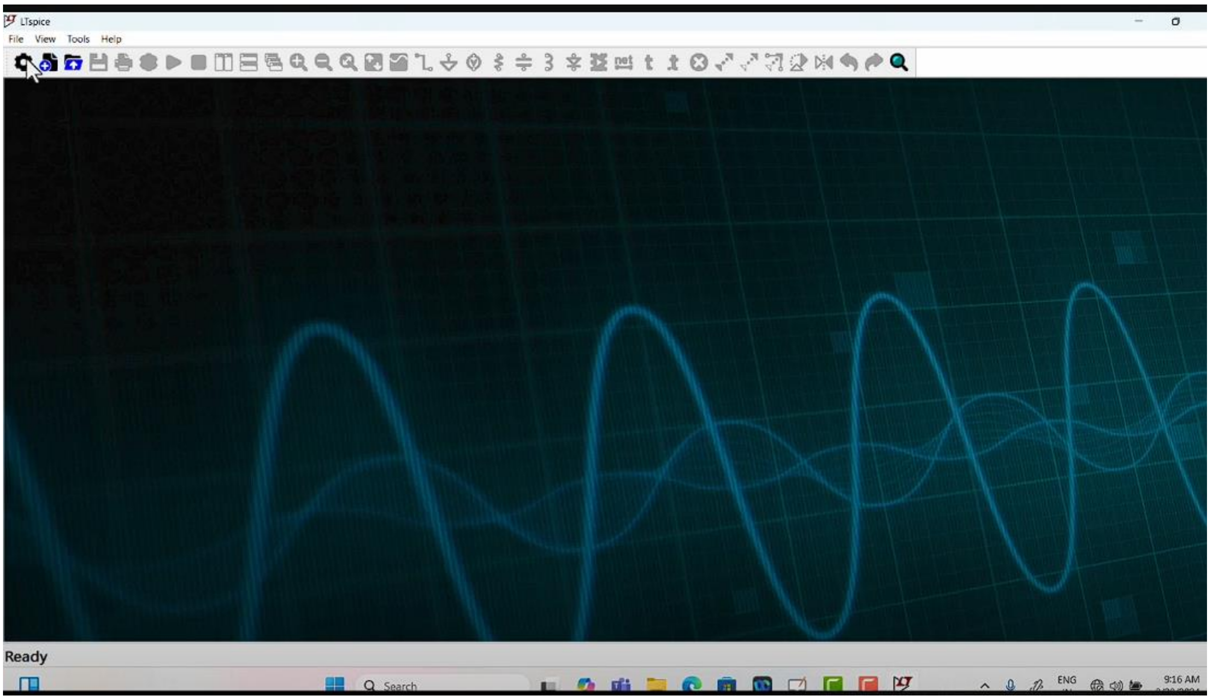
the performance. Then the ripple current and output voltage ripple we need to consider. So, this is considered as 10 percent and 5 percent. So, inductor current ripple and output voltage ripple. Now, when we have all the specification, so this specification either you can just take the same specification just to simulate the same converter. To develop the basic understanding or you can use actual specification to simulate the converter for particular application.

You just choose as per the given specification or as per the requirement of the application. Once you select this then you have to calculate the other parameter in the converter. So here we have all these parameters, all the specifications. By using this, we need to find out what will be the inductor and the capacitor of the boost converter. Because you know as the ratings are given, we know what will be the voltage across switch and the diode of the boost converter.

Because in boost converter, only we need to consider the four parameters. other than the input and output. So, out of four parameters, two are filter parameters that is L and C, other two are the devices. So, one is switch, another is diode. So, either you can use one control, another uncontrolled or maybe you can use two controlled devices, that is up to you, right.

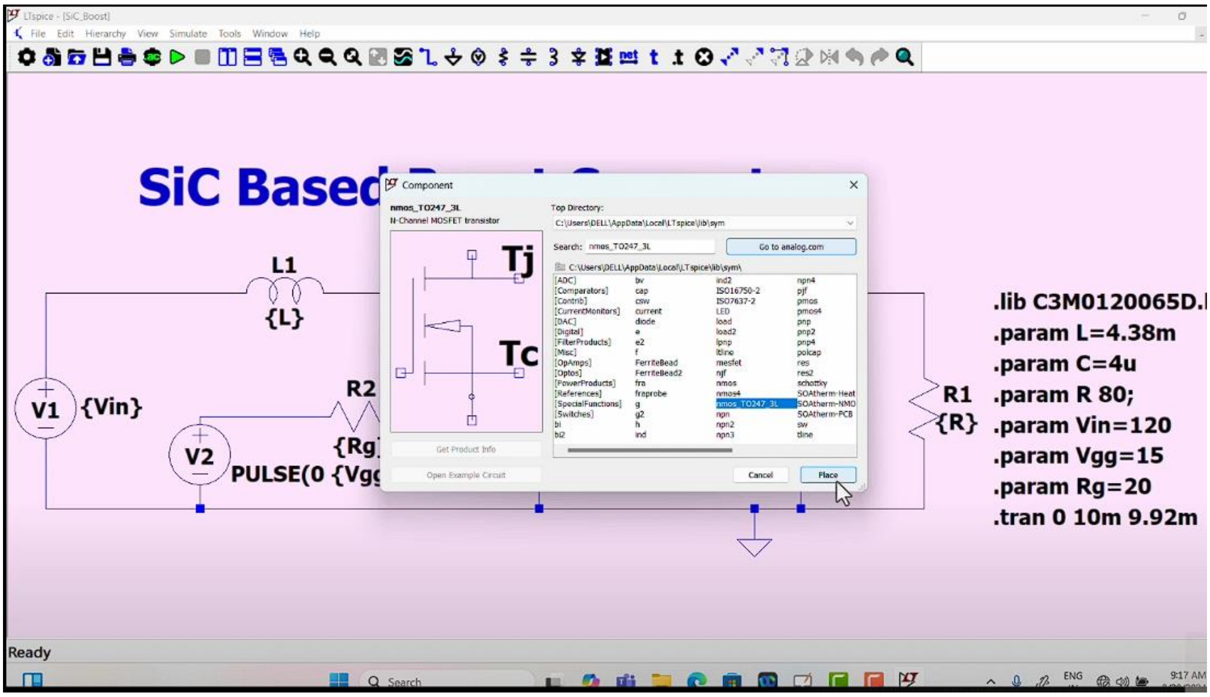
Now, when you have these values, so this calculation you can just follow any power electronics book in order to get these inductance and capacitance values. So, they have given the detailed analysis for this calculation. Any power electronics book you can just refer. Now, by using these different ratings, now we have to build the boost converter. So now let's see how to build the boost converter.

Refer Slide (8:55)



So first we have to go to the LTspice, right? So you can just search the LT spice here. Then here we have already designed the boost converter in LT spice. So we are just showing that boost converter here, right? So, now how to import the device that probably I'll show you.

Refer Slide (10:50)



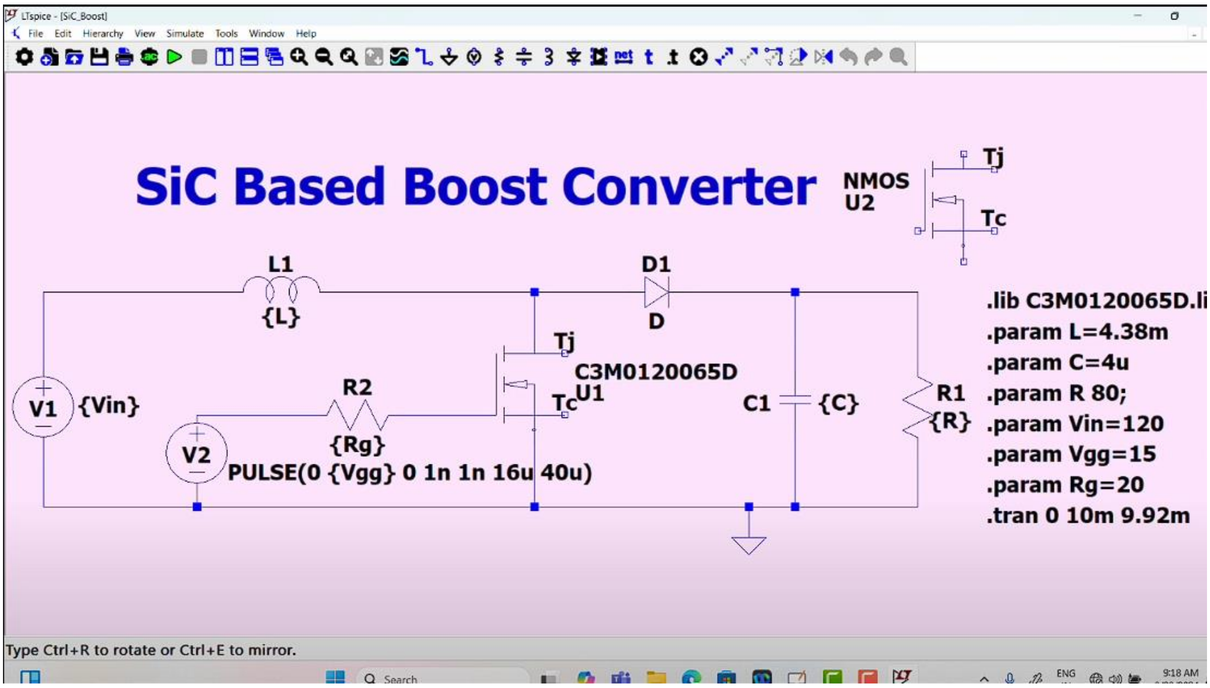
So, let's go to this document. So, basically where this imported device is kept. So, you can see here. So, when you are pressing this directory. So, like components where all the components are present, so, there all the components are present. So, you can see the C, User, then Dell, then AppData, Local, LTSpice. So, under LTSpice, Library and Symbol. So, in this all the components are present.

Now, out of which what component you want to choose. So, whatever device you have imported, you have already kept some name. So, that device directly you can search here. So, that device once you search, so then let us just show you how to search this.

So, see this is some device. So, as in searching part, we are actually writing the name. So, you can see here. So, this nmos TO-247 3L, this is the name we have kept while importing the device. So now this once we are searching it is already showing in the left hand that what the device is. So you can see here so description of the device is already given in the component N channel MOSFET transistor.

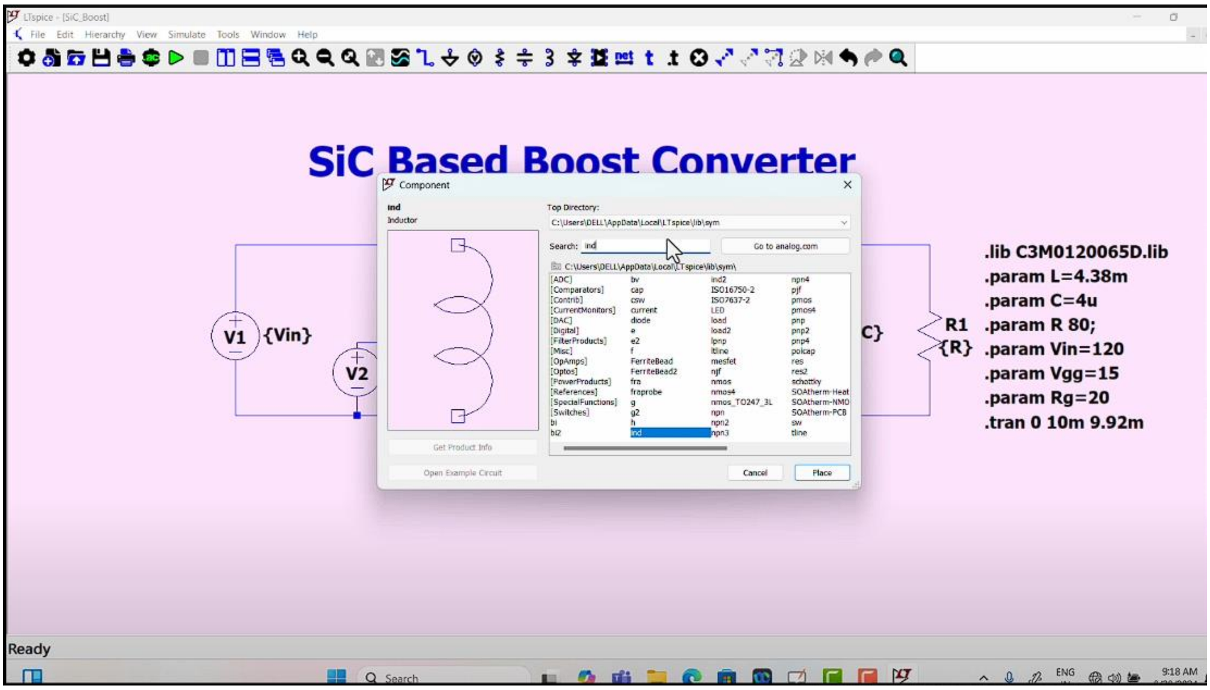
So select this N channel MOSFET transistor you have to just press place. So once you are pressing place so then this will be selected right?

Refer Slide (11:00)



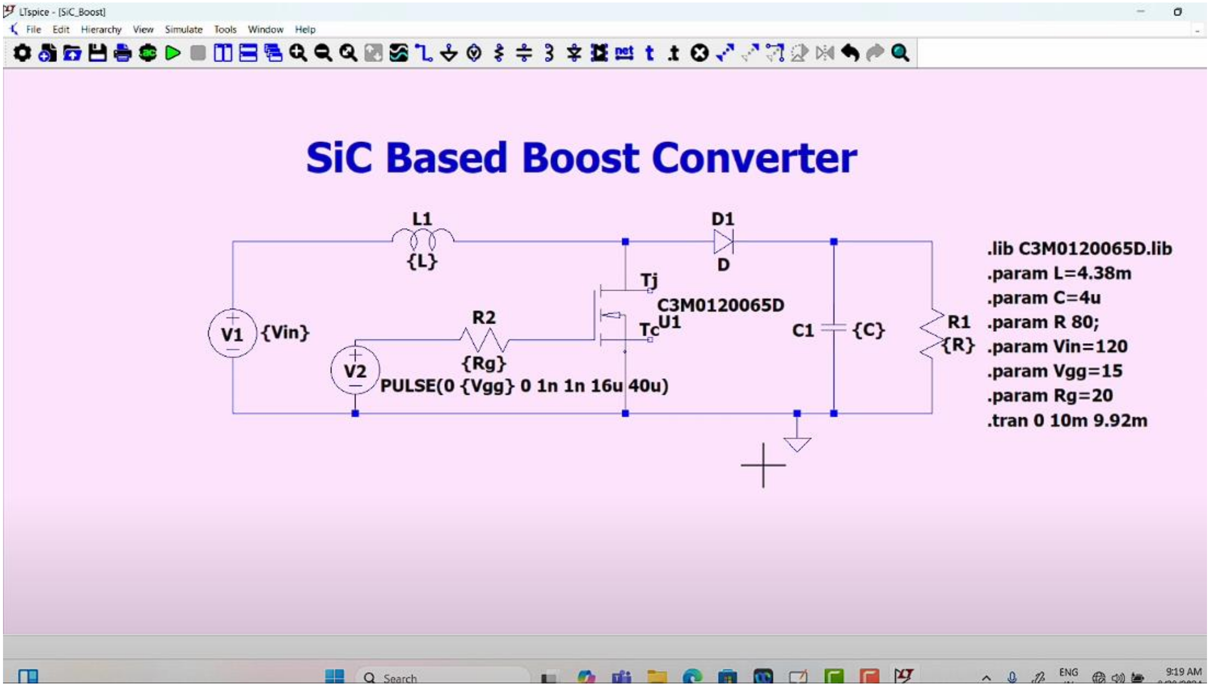
So then this you have to place anywhere. So you can see this is the one which we have placed in the converter. Since now, I don't want to place it. I can just press escape so that this symbol will go. So this okay, let me just escape, and I can delete it. You can if you can place and then you can just delete it, okay? That's it. Now you can see this is the converter,

Refer Slide (11:39)



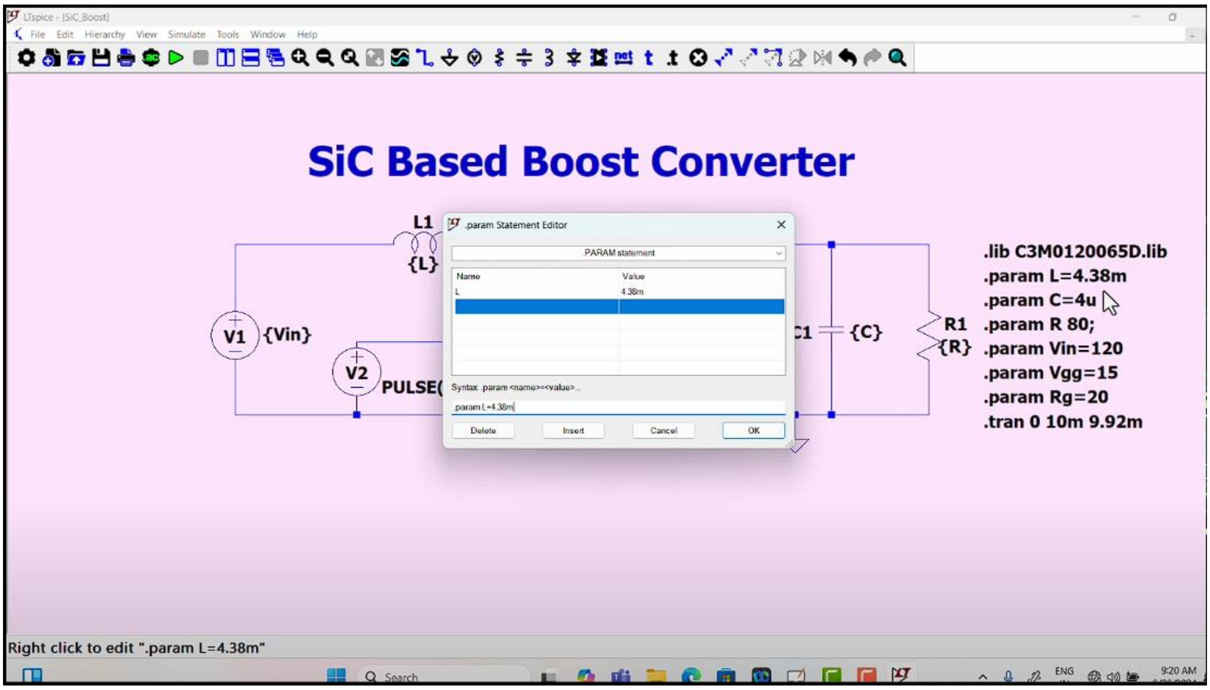
Now if you want to search any inductor you can just search here inductor, so let's write inductor so see this is coming. So similar way if you want to find capacitor you can do that similar way if you want to search diode, if it is not the imported one if it is normal diode you can easily get it. If you are importing some diode let's say you are considering silicon carbide device so maybe you want to use silicon carbide diode so then you can import the diode first then only you can search it here otherwise if you search it directly you will not get the imported device. Right?

Refer Slide (12:15)



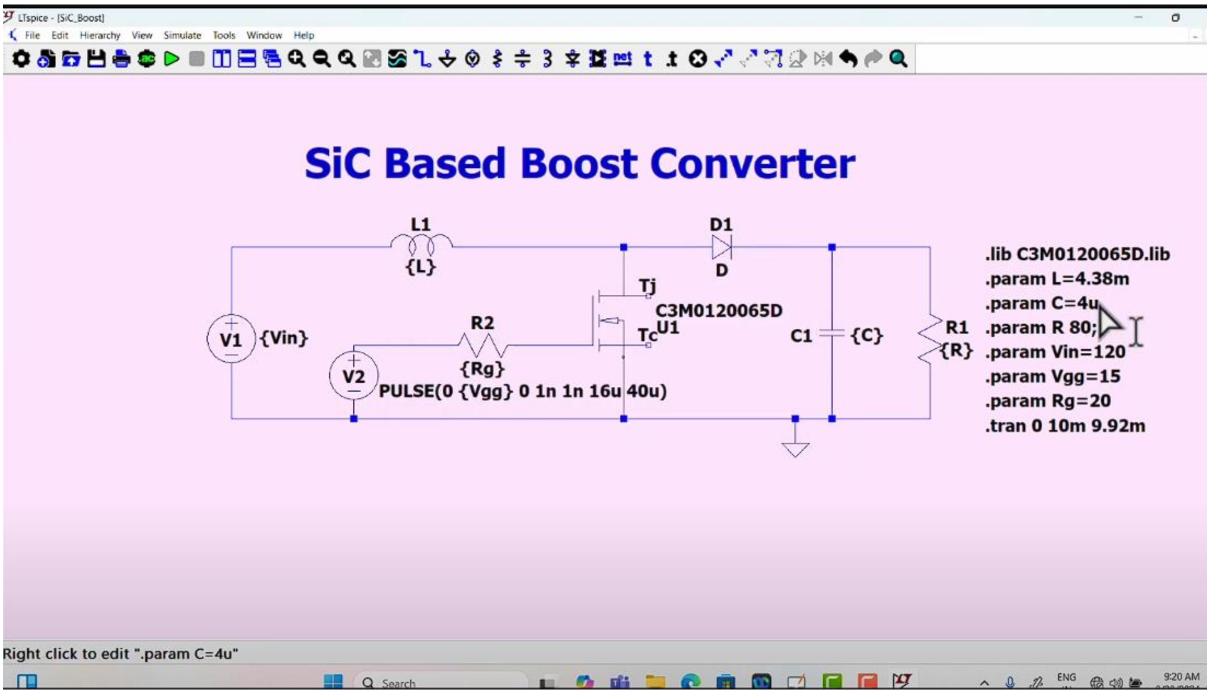
So, I am just cancelling this because you know we already have the boost converter here. So, you can see here so different parameters are given. So, this L, then switch, diode and the capacitors. And we have some specifications of these parameters. Then what we have to do? This parameter you can see in the right hand side, so, L is kept as per the calculated value. So, 4.38 milli Henry. So, you can actually write m for milli Henry. If you are not writing anything then it will take default value as Henry.

Refer Slide (13:07)



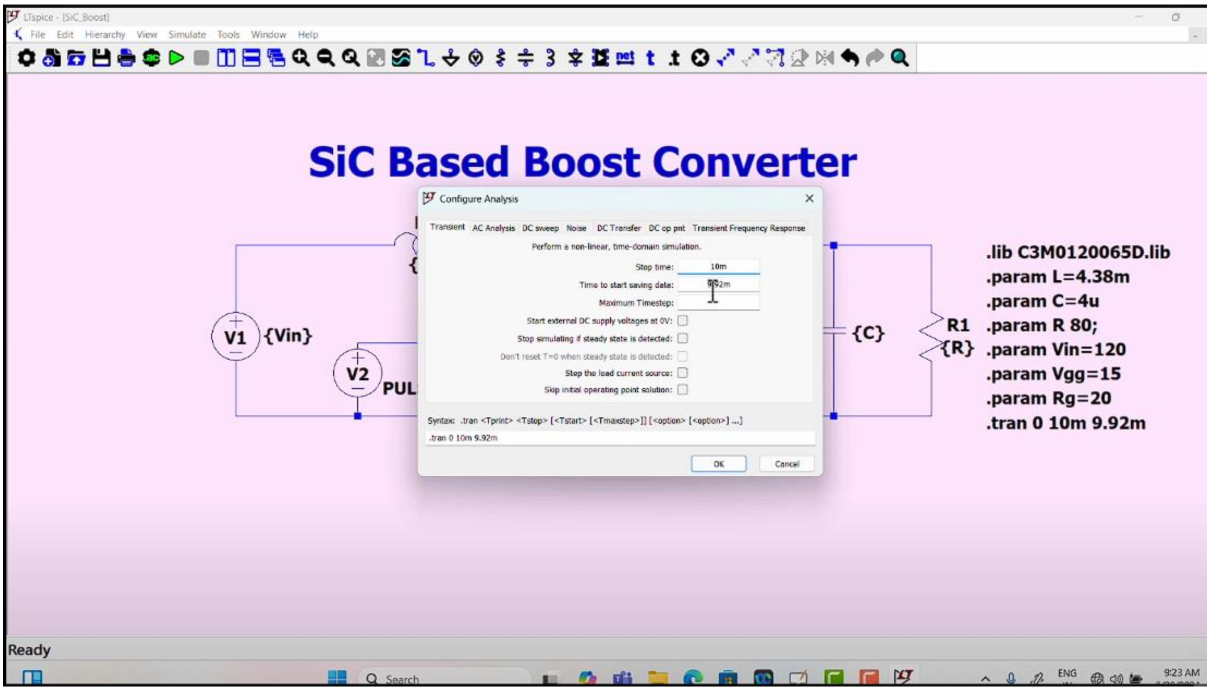
So, in the last lecture I told you that you can define it in default value. I meant that you have to select the particular value and if you are not defining anything else it will take the automatic unit as henry or farad.

Refer Slide (13:22)



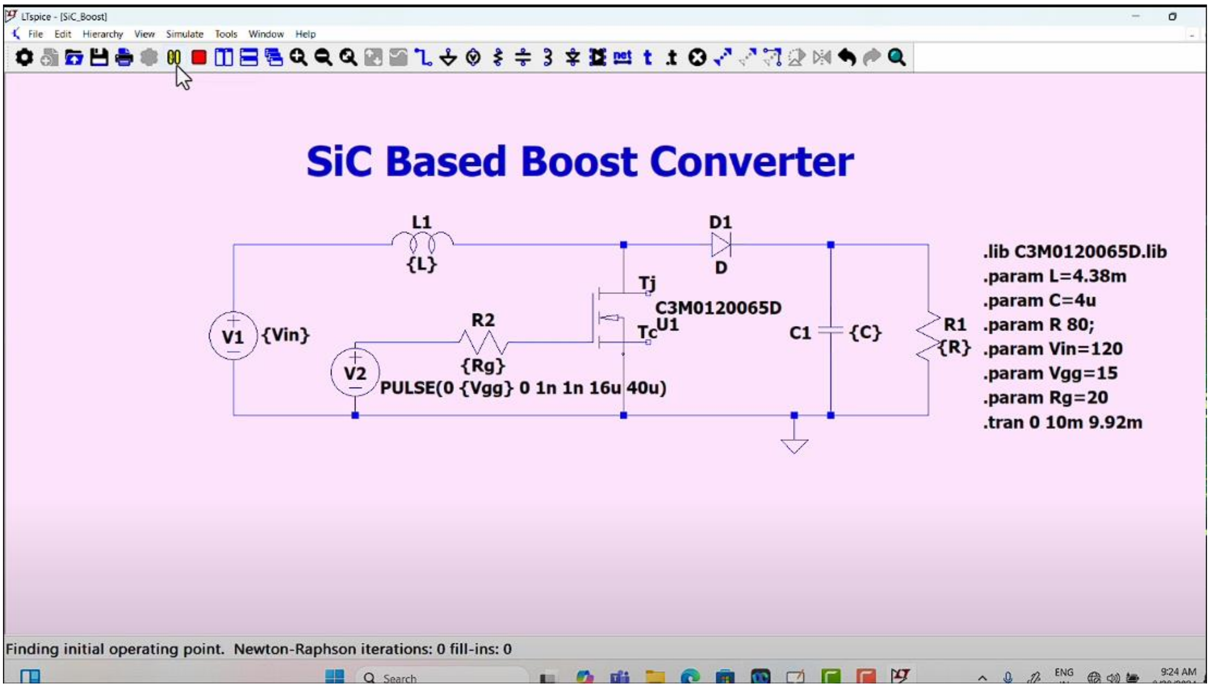
But if you write m then it will consider as milli henry and similarly for capacitor you can see this is kept as 4 micro that means it is 4 micro-farad. So, same way you can actually define the parameter as it is suitable for you. So, in this case the resistance which is considered as the load resistance it is kept as 80 and the input it is given as 120. So, that is the thing is kept here. And then here the important thing is that Vgg, it is the gate voltage. So, this gate voltage you can select as per the requirement of the device. Since we are considering silicon carbide device, so gate voltage is kept 15 here, right? And the parameter for RG, RG is the gate resistance. So, gate resistance is kept 20. So, this is the resistance it is basically the external resistance of the device. So, internal resistance if there is any that is already included in the model because this model we are importing. So, that is the benefit of this particular software since we are importing, so, whatever internal parameters are present in the device as the parasitic parameters, parasitic capacitances or the internal gate resistance that is already included. So, we do not have to worry about that. So, only thing we have to consider is that whatever we want to connect at the outside. So, Rg is the gate resistance with respect to the external gate resistance. It is not considering the internal part, only the external resistance. So, that we are considering here. Now, the transient simulation, so transient part is given in this simulation time. Now, this, so, you can see here this is actually this is the pulse part, so, like what will be the pulse duration because you are considering 25 kilohertz. So, accordingly the pulse you have to give the on off time of the of the switch and also you have to calculate what will be the duty cycle. Already the output is given you can easily calculate the duty cycle and from duty cycle you can easily find out based on the switching frequency what will be the turn on time, how much will be the turn off time and that is what you need to give in the gate pulse okay?

Refer Slide (16:10)



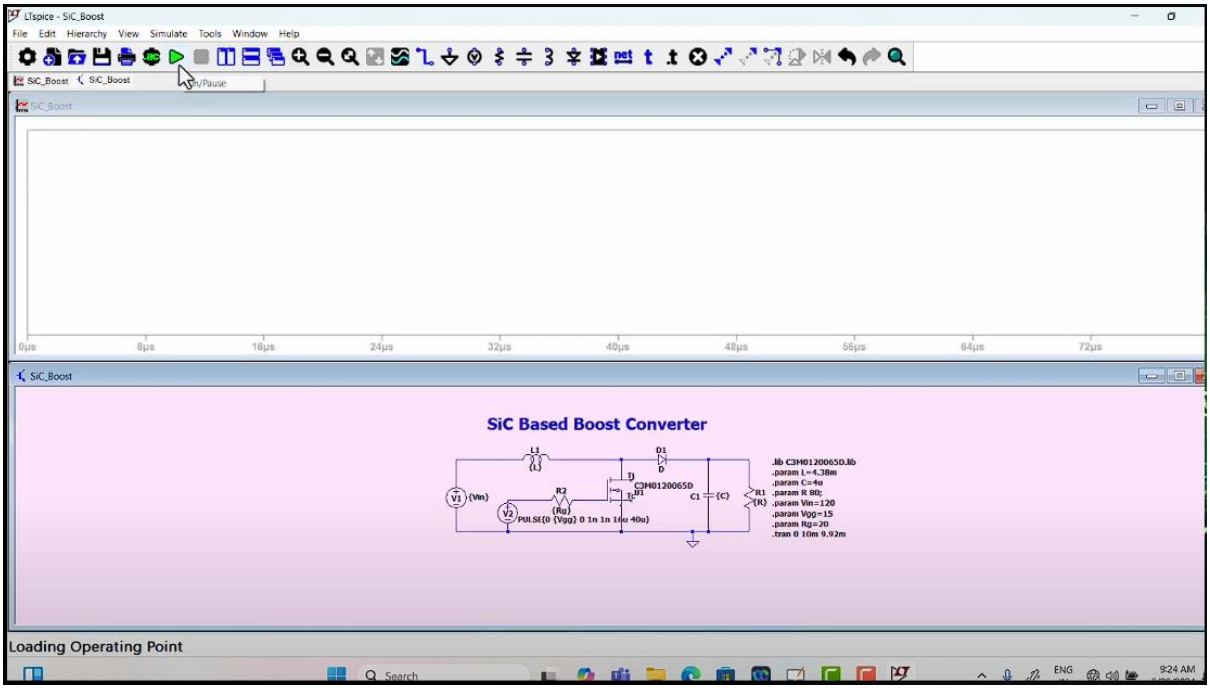
So now, what we need to do we want to simulate this particular converter so when you want to simulate you have to press first you have to actually configure this so basically you can see so we are actually saving the data in last few seconds so basically the complete running time of this converter is 10 minutes. So, 10 milliseconds, it is not minutes. So, whenever we are giving, so that is millisecond. So, 10 millisecond if we are simulating it for, so then out of 10 millisecond, we are saving the data from 9.92 millisecond to 10 millisecond. So, for very short duration, when it is reaching in steady state, we are not considering the initial transient part. Okay?

Refer Slide (17:10)



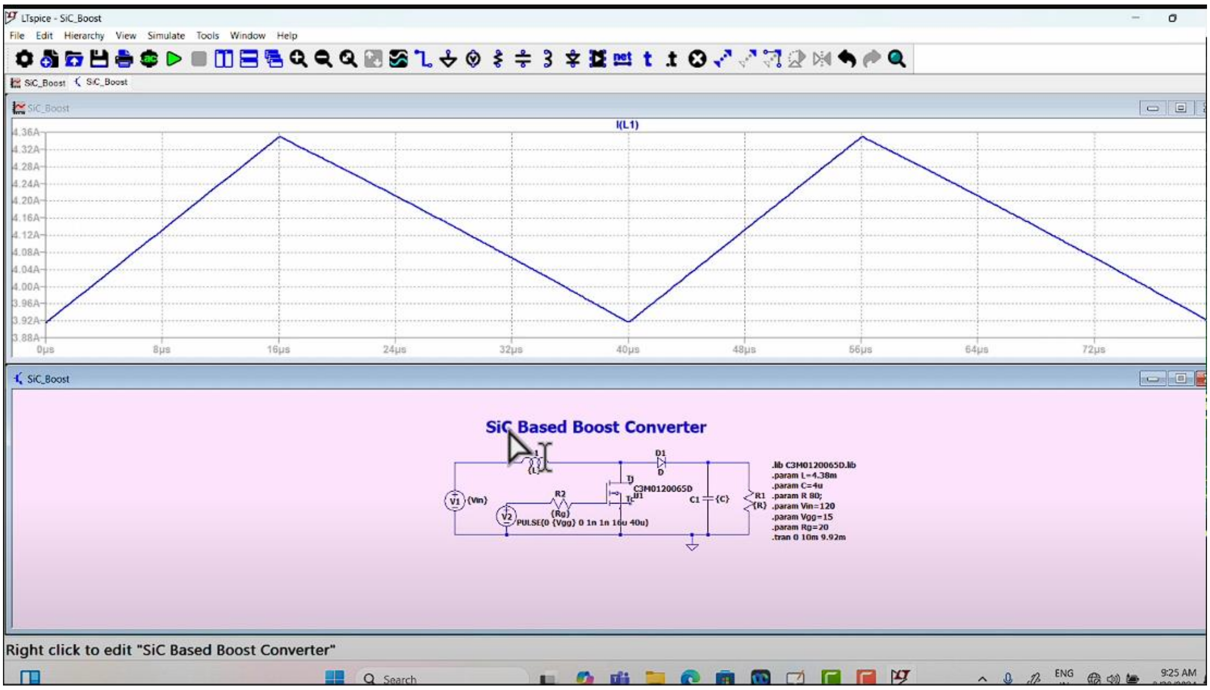
So, now, then we have to run the simulation. Running the simulation, you can see here, this is the part which I already told you, you can run this simulation.

Refer Slide (17:17)



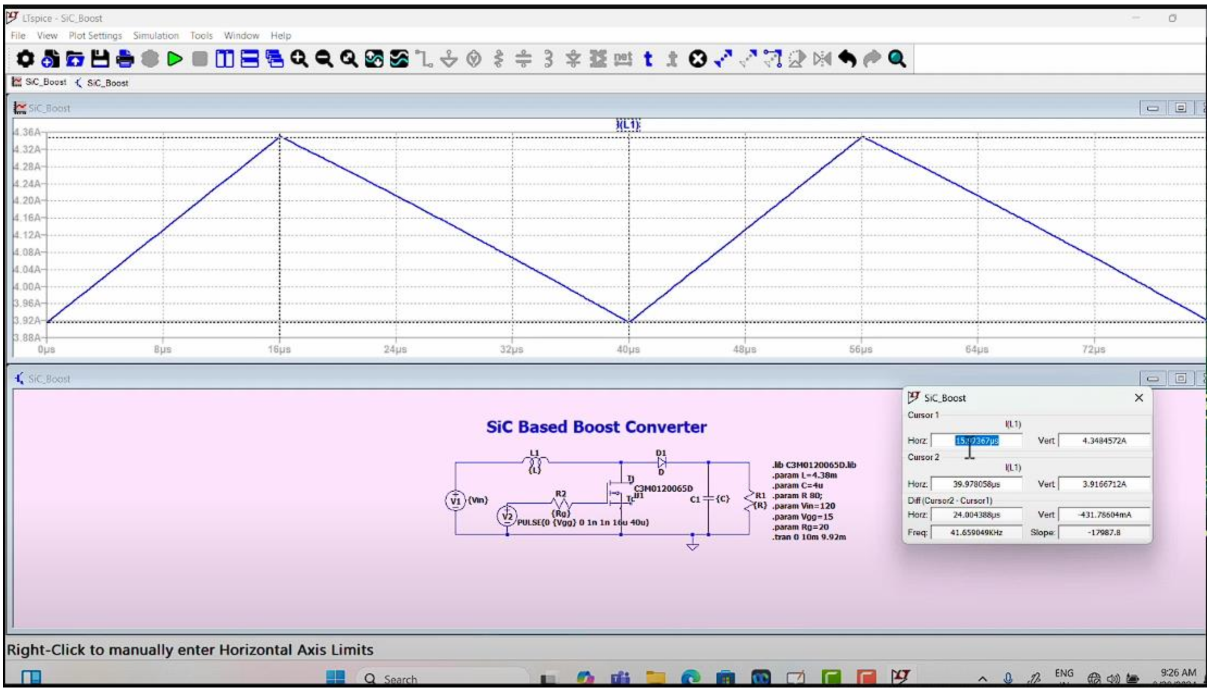
So, in MATLAB and all it takes some time to complete the simulation. Here the simulation is much faster. So, you can see 10 millisecond simulation it is done within few seconds. So, now what we need to do? This is the window so the upper window is where we can see the waveform of the particular part what we want to see. Now what we want to see so let's say we want to see the inductor current.

Refer Slide (17:54)



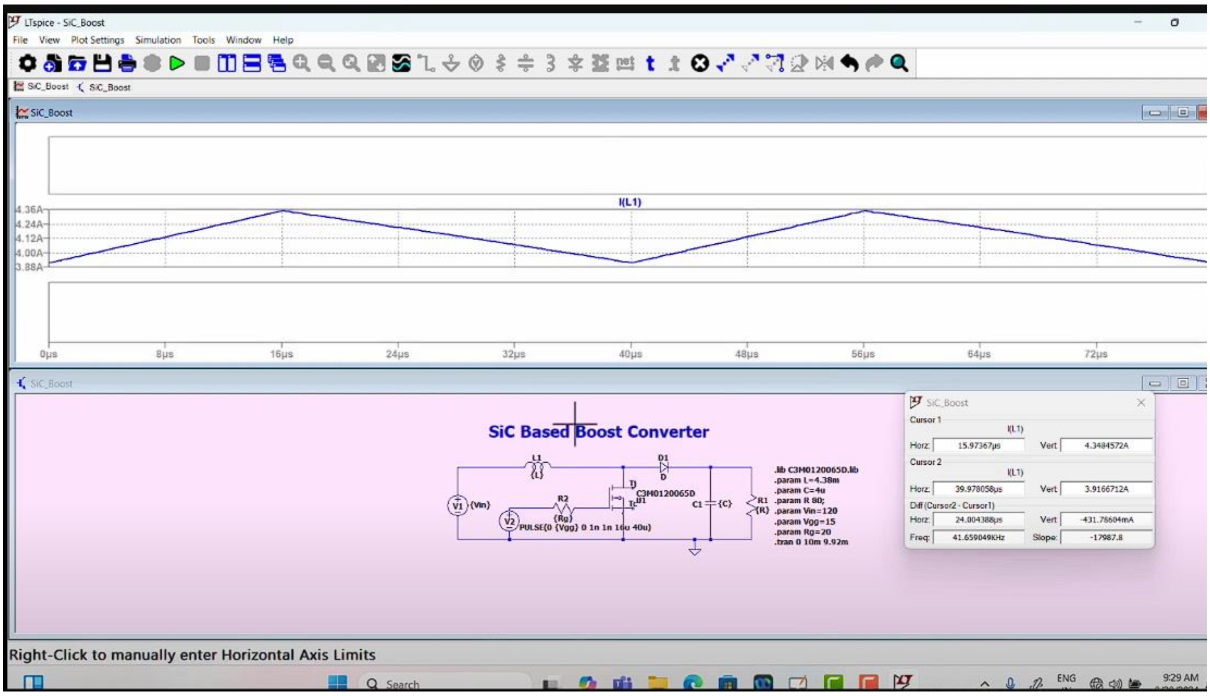
So inductor current you just need to press there. Right so you if you just move your cursor you don't have to connect anything so just move your cursor and go to the place where you want to see the output and then just press it there and then you can see the inductor current so we already have decided how much will be the ripple. So 10 percent ripple it is considered right? Now you can see here this 10% ripple it is visible in the simulation.

Refer Slide (18:51)



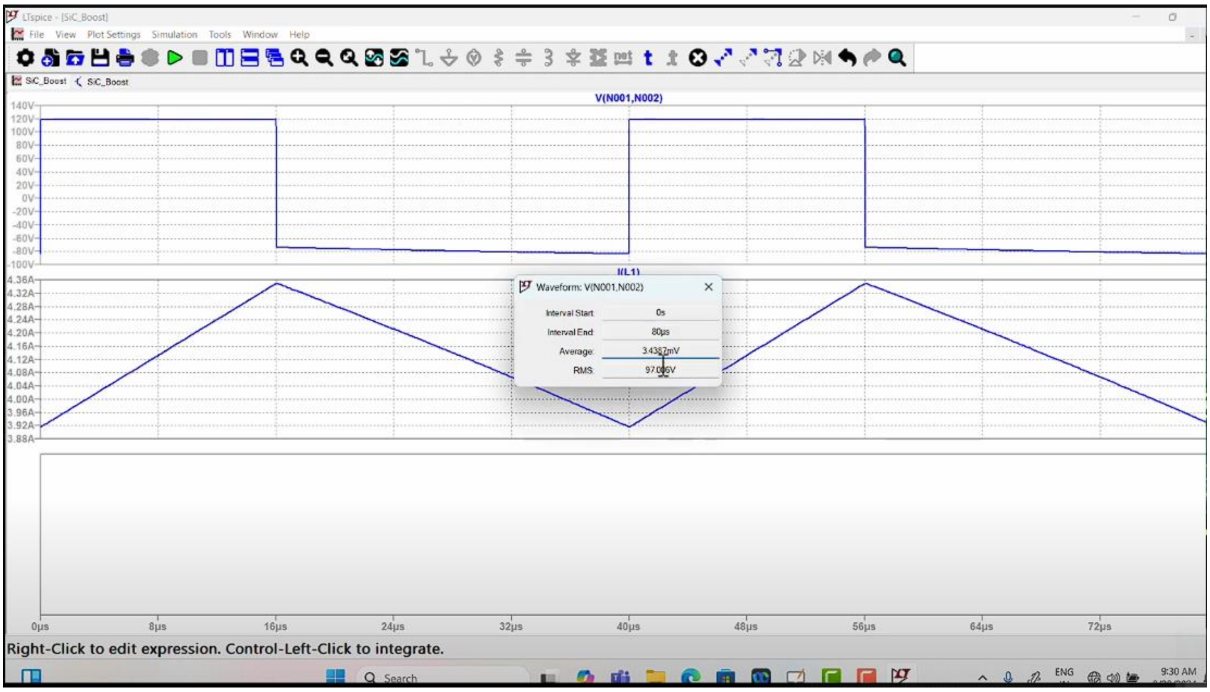
So this is the part you can actually use the cursor. So cursor you can just move and then you can check that what will be the value of the ripple. So, once you are pressing in that point then different cursors are selected. Once the cursors are selected then this window will open, right? So, in this window you can see since two cursors are selected. So, out of two cursor, one cursor what is the horizontal time and what is the vertical time. current magnitude because we are measuring current, so that is visible directly here. And then similarly in cursor 2, what is the horizontal time and what is the vertical magnitude of the current. So, you can see cursor 1 gives the maximum current whatever we can actually consider inductor current. So, inductor current when we are considering that is average current we can consider as IL. Let's say by considering the ripple IL plus delta IL by 2 that will be the IL maximum. So, that IL maximum is the magnitude what is visible in cursor 1 that is 4.34. Similarly, we can get Ia minimum that is the average inductor current Ia minus delta Ia that is the ripple in the current divided by 2. Then it is coming as 3.916. So, this is known, this is actually you can consider as IL minimum. So, IL maximum and IL minimum you are getting. So, you can actually find the difference. Already they have given the difference here. This difference you can see this vertical it is coming around this much milli ampere. So this difference automatically it is calculated in this. It is so simple. It is very easy to actually see this. And then so this difference you can see here. So this frequency it will be it is around this and the slope it is given here. Right? Frequency this means like you can actually understand so basically the entire thing it will be the half of the part we are actually seeing so that is why the frequency you can see here it is almost like double it should be double but if we see the entire cycle when the starting it is starting from here and then it will be like this then it will be 25 kilohertz which we have selected. So this is very easy to actually consider. So this you can see here the average current you can actually you can just press in this and then average current you can get this is 4.1358 ampere and then you can also get rms current so if it is ac then obviously there will be some meaning of seeing rms but it is dc so obviously it will be similar. Then interval start 0 second interval end you can see here 80 microsecond. So the complete information you will get for any particular parameter.

Refer Slide (22:20)



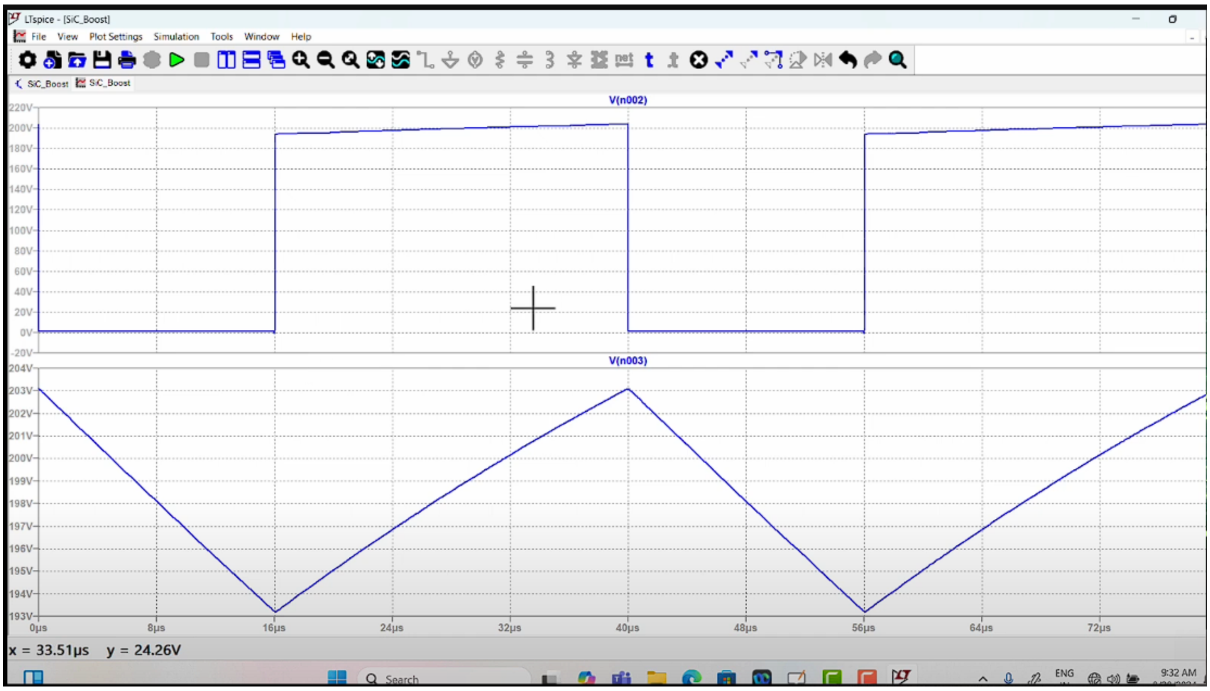
Now this is with respect to the current. Similar way we can see the voltage also. So let's just connect the voltage probe in the inductor. So, this when you are selecting, so it is coming just above it. So, you can just select this, add plot pane below. So, then add plot pane up. So, then what will happen? So, there only it will be connected. Right? So, now if you are selecting the inductor voltage, so you just need to press here. So now you can see this voltage. So then you will get so one probe if you connect then obviously it will show you the voltage with respect to ground that is the input voltage. So that is why it was 120 volts. Now if you want to see across the inductor then what you have to do you have to actually connect this this red pin in both side of the inductor. Then it will give you the voltage across the inductor. Otherwise, one pin will give you voltage with respect to the ground can see here so this is like the by default thing so then here you can see this voltage across inductor it is it is visible here so you can see for during the charging it is positive and during discharging it is negative.

Refer Slide (22:31)



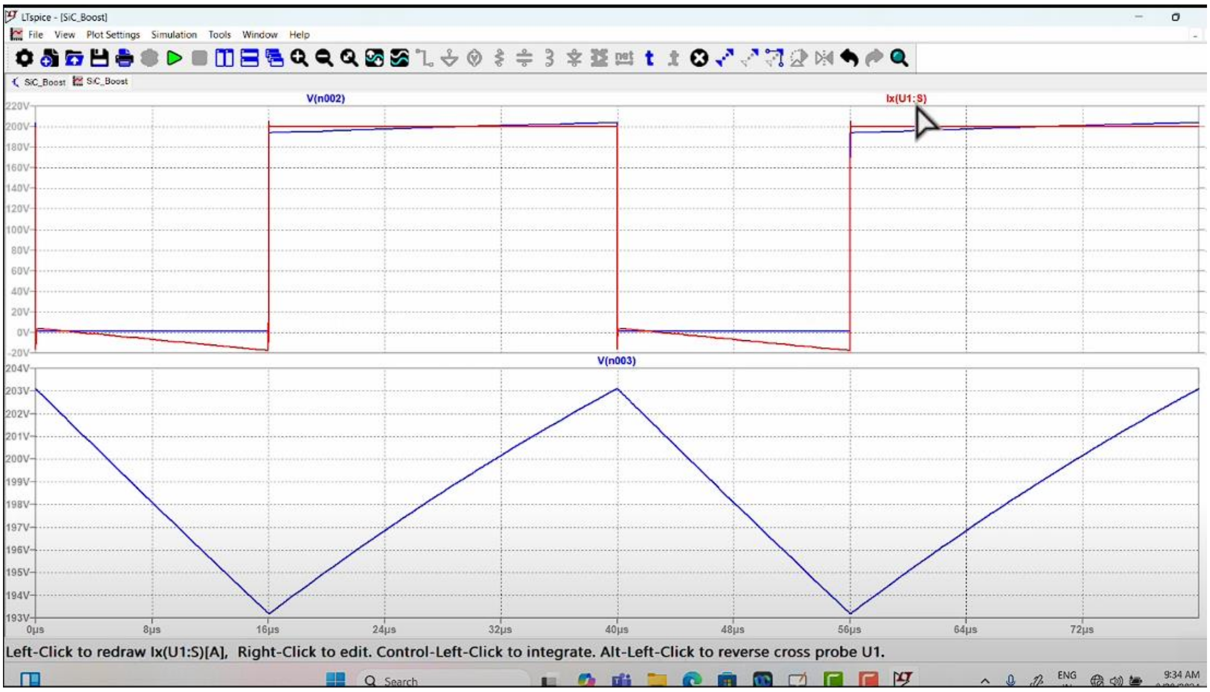
Now it is visible clearly to you so you can see here this is the positive part and then negative part and average voltage of the inductor will be zero. So, this you can calculate. So, average is 3.4387 millivolt. So, that is like close to 0. Since it is considering some parasitic and all, it is having some voltage. So, millivolt is like close to 0. So, you can now see the RMS voltage. So, since this is having positive and negative, it is kind of AC. So, here you can see the RMS voltage, how much it is in the inductor. Okay? So now you can see both inductor voltage and current together. Similarly we can see switch voltage and current. So let's see how this switch voltage and current looks like. So you just need to connect these two probes across the switch. We can just close this window this then what will happen it will be easier for you to see. We can just ok so let me just run it again.

Refer Slide (25:16)



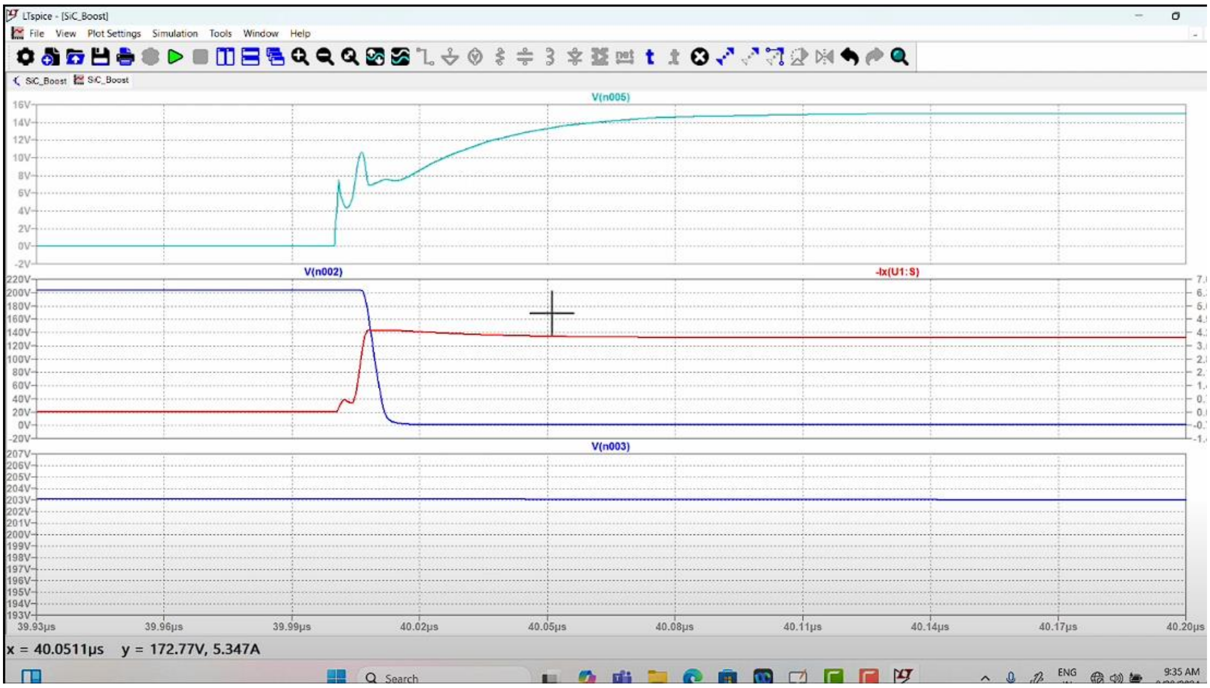
We can just, either you can just use the same window or you can just run it again to see the different part. Okay? Now you can see this is actually the, this is the voltage across switch. Now you can see this is voltage across switch the first one and the bottom one is the voltage at the output. So output voltage is desirable 200 volts. So you can see here we are getting 198.3 volts. It is not exactly 200 volts. This is the non-ideality which is considered in the simulation. Because you know like the parameters which we have considered this is having the non-ideal factor means the parasitic component and all and the efficiency is considered 95%. So, accordingly different part of the converter is operating and it is giving it is desirable 200 but actual circuit also we will not get 200 volts exactly. There will be small differences let us say 2 to 5 percent differences whatever differences will not cause big problem in the output. So, that much differences we have to consider as the error can be present. So, this much is present here 198.3 volts. So, and then switch voltage you can see here. So, now I will just show you switch current along with switch voltage. So, just give me some time.

Refer Slide (26:51)



So, I will just add this. So, now you can see this switch current. So, switch voltage was there in the blue color and the red this is the switch current. So, you can see when the Switch. voltage is there. So, this blue is the voltage and red is the current. You can see here this voltage V is given in blue and I current of the switch is given in red. So, whenever this voltage across switch is present, so that time current is 0. So, this is complementary as the basic principle of the switch. So, this is blue and this red part is 0 and whenever the current is present, so then that time the voltage across the device it is zero. So, just I will just zoom the switching part. So, turn on part I will just zoom and show you that what is happening there.

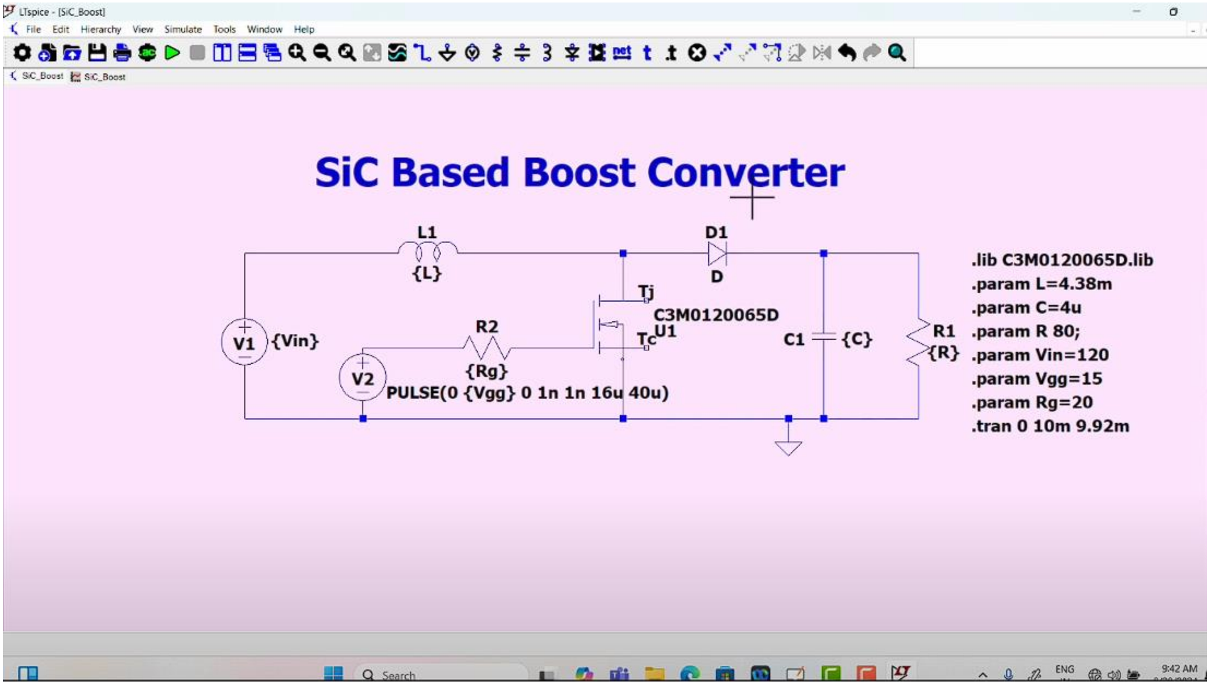
Refer Slide (28:17)



So, you can see this is the gate voltage which is added in this. So, then you can see here this upper one is the gate voltage right? V which is shown in this green color and then middle one where two plots are shown, this blue and the red, this is the gate voltage and current of the switches. Gate voltage you can see here some non-linearity is coming so generally what happens in any ideal kind of simulation this kind of non-linearity is not visible. So gate voltage generally it turns on immediately when the voltage is applied because you know parasitic components are not coming into picture but here since it is coming into picture so you can see some oscillation across the gate switch so this is the point so this voltage is across gate and source of the switch not from the gate voltage where we are giving the pulse. It is across the switch where this input capacitance of the switch is corrected right? So there this oscillation is coming so it is taking some time to reach the final voltage. So, basically you can see here, so this is the point here actually it has reached to the final gate voltage or the voltage which we are desiring. So, you can see here, so this is the point it is 14.19. It is not 15. So, what voltage we have given? That is 15 volts. So, you can actually move your cursor to see where exactly this 15 voltage is reached. So, you can see here. So, this is the point where the 15 voltage is reached. reach. So, then you can see the cursor point. So, this is the point cursor and the another cursor if I connect at the point when the gate voltage is starting. So, let me just connect another cursor. At this particular point so the gate voltage is starting here. So you can see so this is the cursor 1 the first cursor 1 is connected to the point where gate voltage is actually starting and cursor 2 is connected to the point when gate voltage is reached to the rated voltage. So, horizontally you can see this cursor 1 time is given and this vertical means like magnitude it is 20.5 right. And then cursor 2, courser 2, you can see this time is given and vertical the magnitude is given. So, this vertical magnitude for cursor 1, 20.5 microvolt, you can consider that as 0. It is not actual 0, but this is microvolt is close to 0 obviously, right? So, then that is 0 and this is 15. So, the difference in time you can see here, the difference in cursor 2 and cursor 1 is 157.48 nanosecond. So, that much time it is taking to reach the rated gate voltage right? Now if we see the part where exactly the voltage and

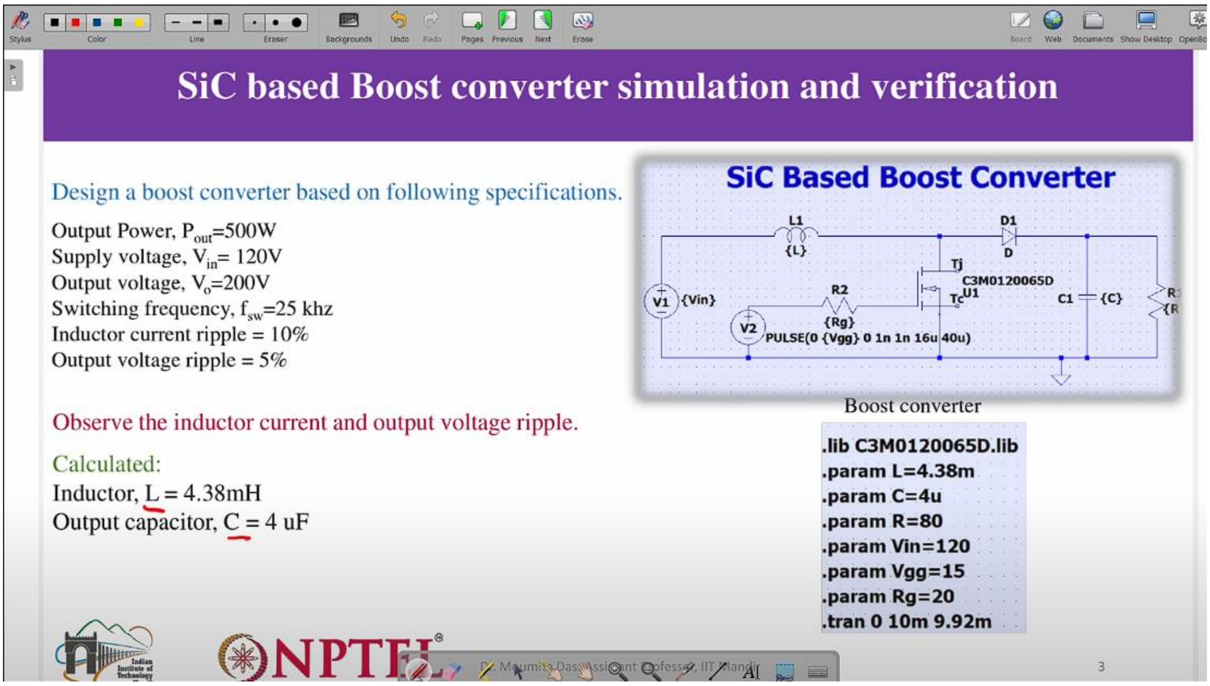
current transients are coming so you can see initially the current is starting at the same point, so there will be some delay point at the cursor one so current is reaching to the actual value at this particular point so you can see here so if i just move this cursor to cursor two. If we just move the cursor to the left side Little bit more left side and just move to the right this point this point here it looks it has reached to the steady state point and this is the time it is taking 79 nanosecond so this is the time this is the on time this is the switching loss which will happen during on time it will be in this particular duration you can see voltage is rising and current voltage is falling current and current is rising there is a clear intersection.
There will be losses and the current is taking some time to settle down, so that time there will be initial transient, the switch will take this much time to settle down to the actual rated value. Now, initially the time between cursor 1 and the first box you can see here. So, basically the point where this first box is coming. So, basically I just show you here. This is the point, so you can see here, so 19.8 nanosecond in this actually box you can see here. So, this much time you can see it looks like there will be losses because you know complete intersection will be there. So, this is the losses will be there during the turn on time. So, now you can use the same window like similar type of window to understand the switching transient. This is not possible to understand in the ideal simulation software. So, this is possible in LTSpice because we are considering the actual switch and you will get the actual kind of transient which will be visible in the practical experimentation. So, you can actually calculate the losses and then check, these losses in the analysis because in analysis we will get some losses during turn on and turn off time whether it is matching with that losses or not. You will find some difference because analytical will give ideal kind of result means the losses will be even the losses whatever we will be calculating that will not be able to capture the actual time. Okay? so this is possible in the simulation actual simulation and then if you see there will be some differences will be there it will not be exactly same and the practical experimentation when you will be performing that time you can check this will be close to that value. So this is the benefit of using this particular software, so you can also like go through other parts of the of the simulation so basically now just close this window

Refer Slide (34:47)



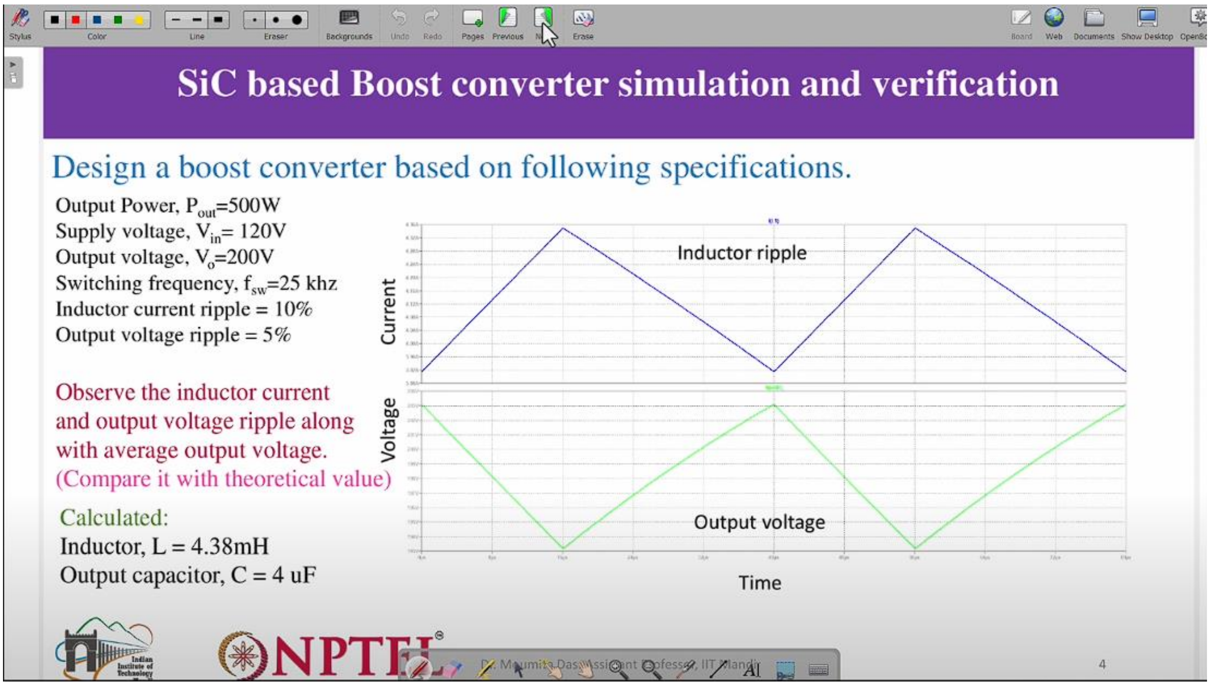
So, you know inductor current and switch is the part where actually we have interest. So, we are actually looking to this part only. So, now if you have diode which is also like silicon carbide diode, you may also see the diode current at the voltage and which will give you the idea about the losses in the diode itself. And then the capacitor anyway it is providing the output voltage. You can check whatever ripple you have considered whether that much ripple is visible or not now. You can also calculate the efficiency. So let's see that the efficiency how much you can get in this.

Refer Slide (35:24)



So you can see here so this is the simulation we have considered so efficiency in this case is considered as 95 percent, right? So, this much you can actually calculate. So, we can just check this input voltage and current in the simulation, output voltage and current and that much efficiency is visible or not that you can consider.

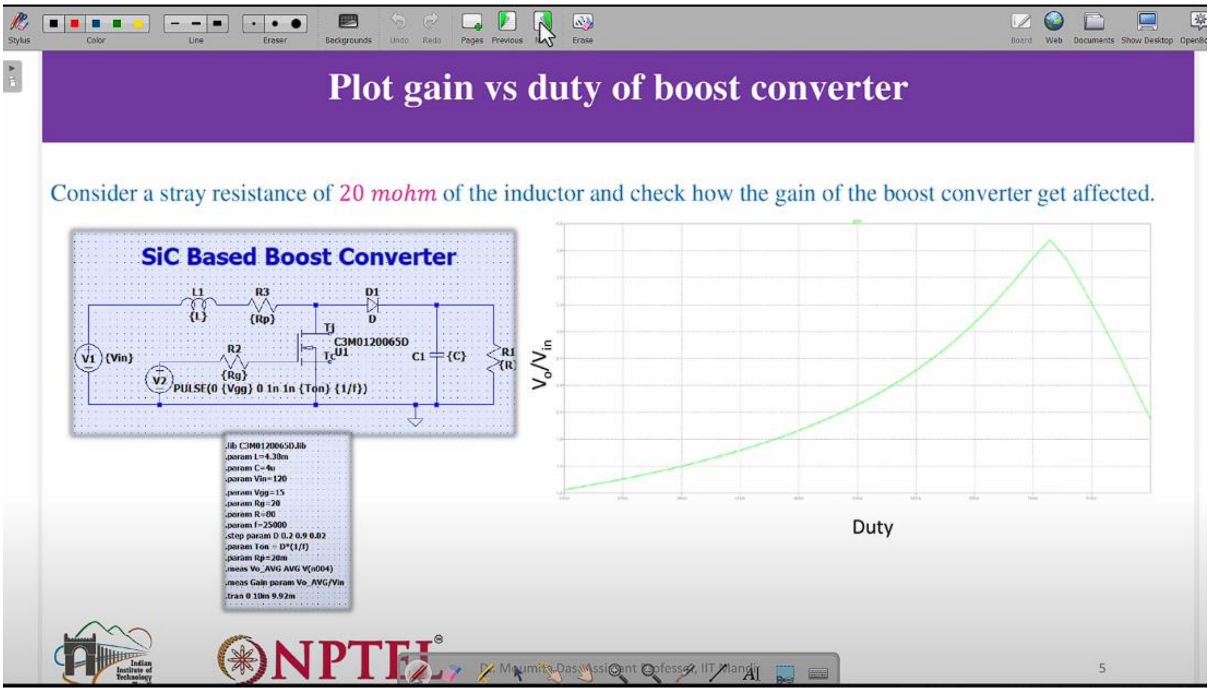
Refer Slide (35:44)



So, this is the part which I have showed you in the simulation itself. So, inductor ripple current at the output voltage, output voltage ripple and the inductor current ripple this is something which we have actually considered.

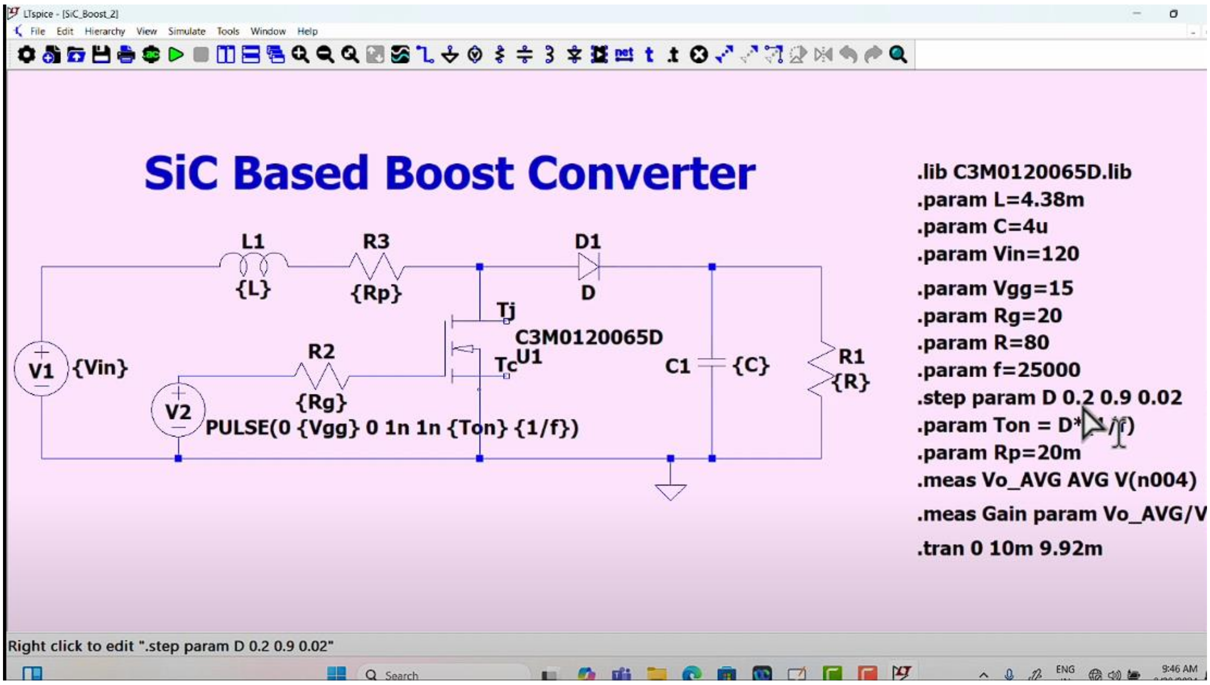
So, this is something which will be similar. So, see here in this inductor current ripple is considered as 10 percent right? and the output voltage ripple which is the capacitor voltage ripple. Now if you want to reduce this 10 percent then inductor size will increase if you want to reduce this five percent then the capacitor size will increase. Now you can reduce the values of inductor sorry, inductor and the capacitor by increasing this ripple. Again this is up to you how much you want to keep for your converter and this is like based on the designer so like this ripple can be variable as much like it will not create any difference in the circuit performance. Okay? Now the inductor current and the output voltage ripple along with the average output voltage it is shown here. You can actually compare this with the theoretical value. That is the first thing you should be doing. Whether the simulation result is matching with the theoretical calculation or not, right. Theoretical value once you are calculating then you can get this all this like matching or if there is a mismatch that will be due to the practical circuit operation.

Refer Slide (37:23)



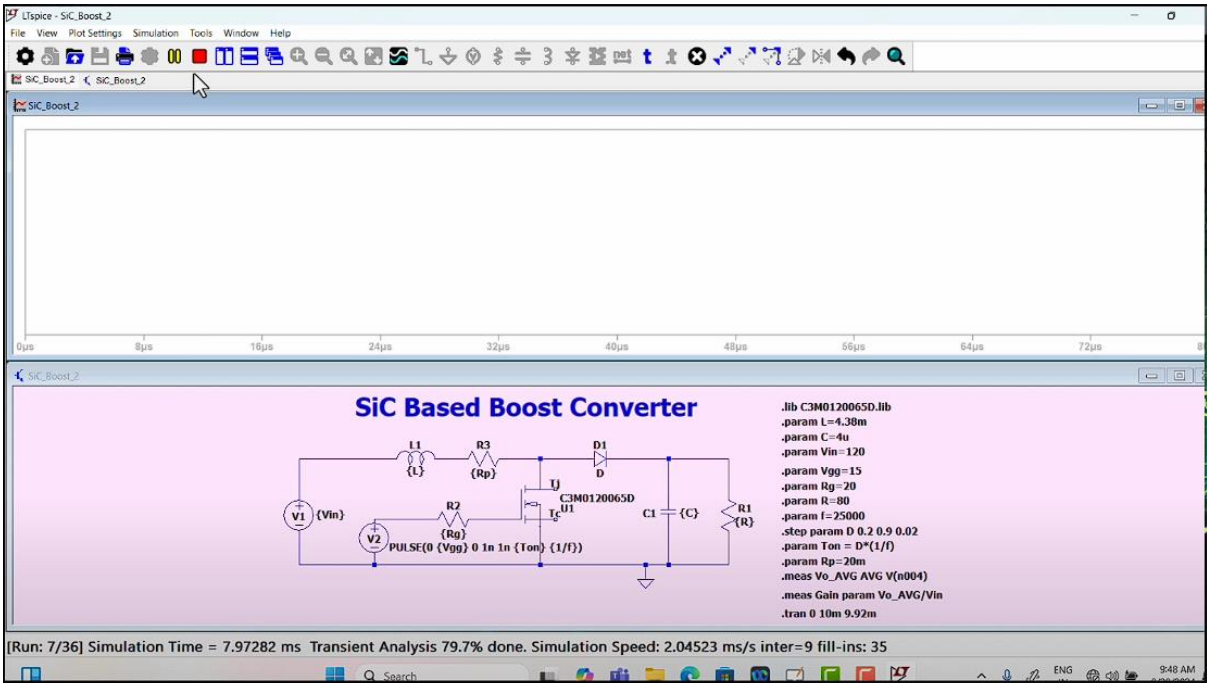
Now what you can do is that you can actually plot gain versus duty in boost converter. How that is possible? So that I will just show you in the boost converter simulation itself. So how this gain will be possible. So, whenever you have to provide this gain versus duty cycle, these are the all things you have already given in the simulation, right? So, now what you have to do?

Refer Slide (38:35)



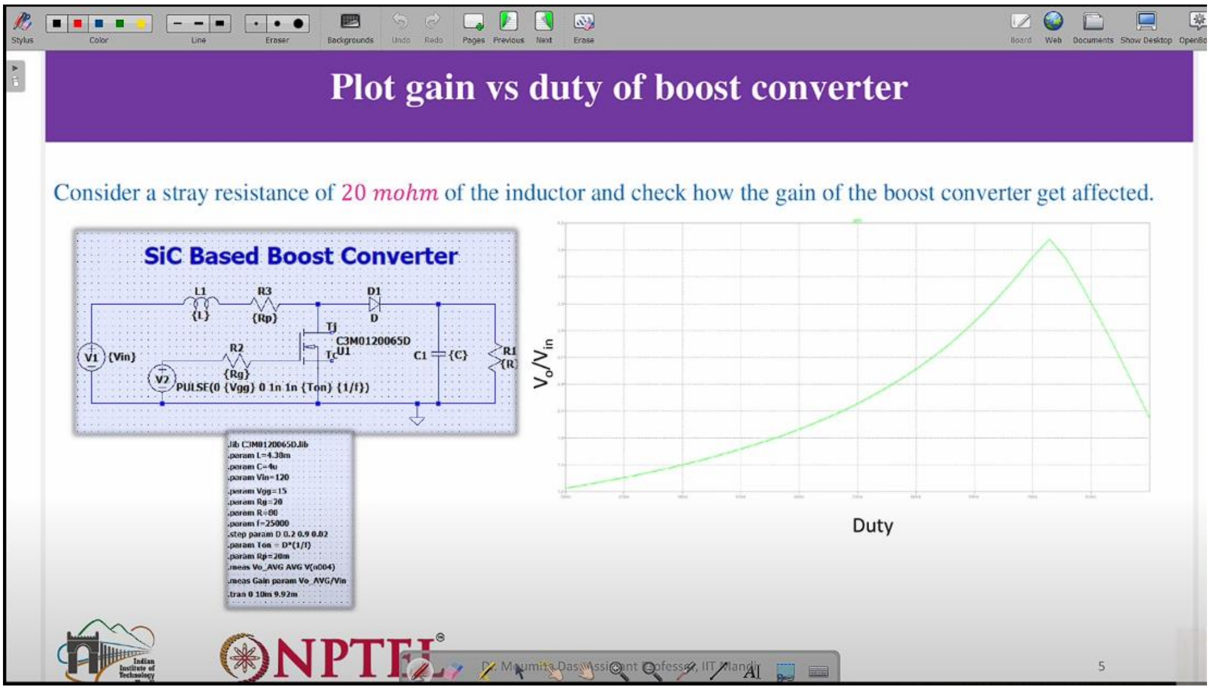
In order to find out gain, so here one parameter is even defined here, the gain parameter. So, what is this gain parameter? So, gain parameter it is V0 average because you know it is having V0 is having some ripple. So, that is why we need to use V0 average to get the average value divided by Vin that is the input voltage this is gain which we want to see and then with respect to the duty cycle. So duty cycle it is actually given here step parameter it is given here so you can see here in this point step parameter of the duty cycle is given first is given 0.2 to 0.9, means the minimum value of the duty cycle is kept as 0.2, you can also choose 0.1 or 0.05 whatever minimum value you want to select and maximum is kept 0.9. You can also select 0.95 or 0.8 Now the steps are given 0.02 like from 0.2 it will vary 0.0 0.22 then it will go to 0.24 like that steps will be coming and each step gain will be calculated and then this gain will be plotted. So, let's see how this gain plot looks like. So, we will just run the simulation here again. So, you can see the same this configuration you have to do every time when you are simulating any circuit. So, the same start and end time we have and the same time to save the data is same as the previous simulation which I have showed to you. Now, just we need to simulate it.

Refer Slide (40:15)



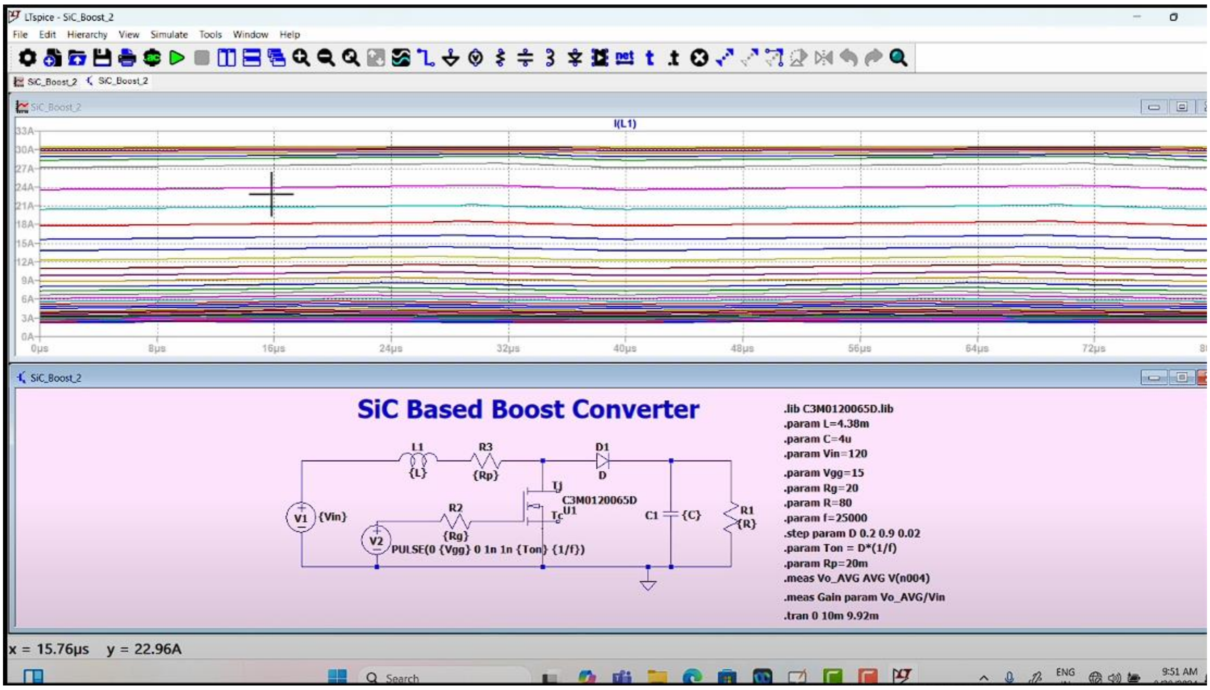
So, now we have this. This is still running so you can see it is like yellow means like this is still running, so if you want to stop in between you can just press in the red highlighted this notation, so then it will be stopped but it is still running because you know it is calculating so you can see here because we have some calculation also added here so it is taking some time the previous simulation it was fast immediately the simulation when you started then the simulation immediately it gave the result. But here since this gain parameter we have and the duty cycle steps which step wise we are actually changing right. See it is taking some time and the time simulation speed is given here and you can see at the bottom simulation time where it has reached ok? And then transient analysis how much percentage of the analysis it is done. So, this is also showing and the simulation speed and the inter different interval it is showing here. So we just need to wait for some time. So sometime probably the simulation will take some time. That doesn't mean that there is something wrong. But if the simulation is more complicated or maybe it is having some more analysis part or some more points you have added. So then it may take some time. You just have to wait until the simulation is completed. So in the meantime you can do the analysis in parallel to that by yourself. So, I will just in the mean time I will just show you this waveform how it will look like. So, it will look like this. So, you can see here as the duty cycle.

Refer Slide (41:55)



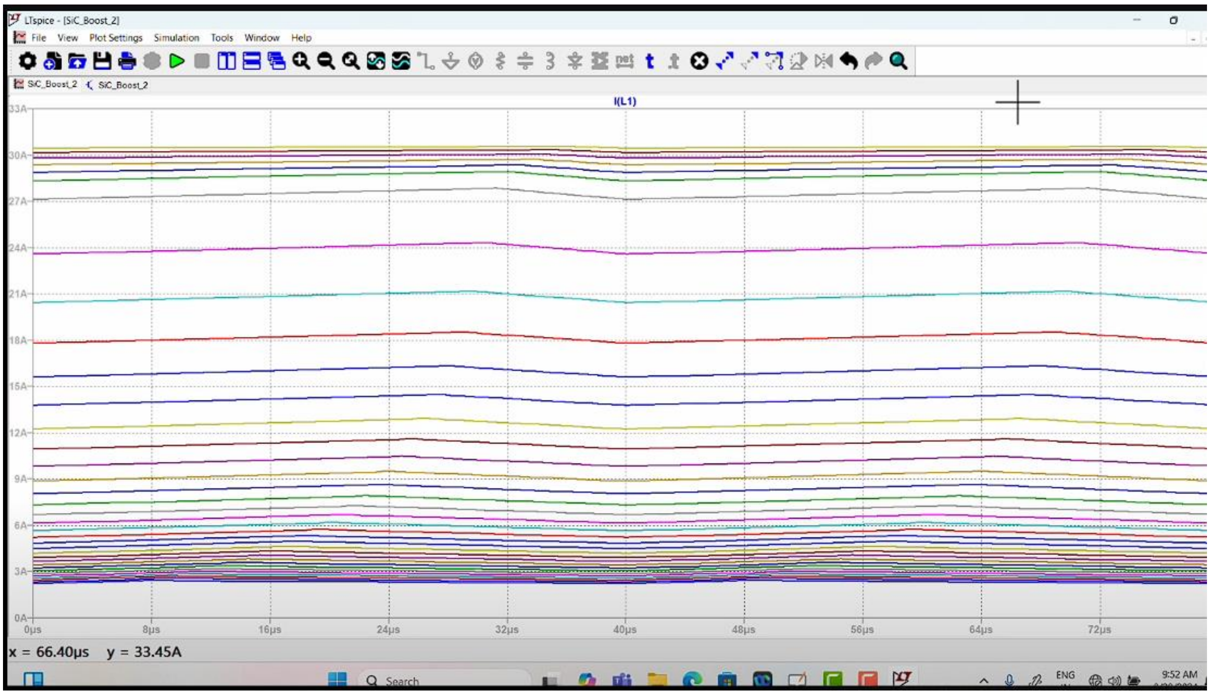
So, the x axis is showing the duty cycle. So, duty cycle is varying from 0.2 to 0.9 and then V0 by Vin is also variable. So, till some duty cycle this is increasing. So, you can see till this point it is increasing linearly. Right? After that point even if we are increasing the duty cycle, then the gain is not increasing, then the gain is falling. So, this is from the basic of the boost converter, you cannot get infinite gain at 100 percent duty cycle. Right? So, this is the same thing it is visible here. But if it is like ideal kind of circuit, it is considering all the factor. If it is ideal circuit, then probably it will show you infinite gain. Because the non-idealitry factors are included, so that is why it is showing the waveform similar to the practical circuit. In practical circuit also you will not get infinite gain. After some duty cycle the gain will fall. Till some duty cycle only the boost converter will be boosting the voltage, input voltage. After that the voltage at the output it will be falling. So, this is the same thing you can see in this particular simulation. So, this is just to give you idea that it will be more close to the experimental circuit. So, the output will be similar to that. But if you analyze also, analysis will give you the infinite gain, right? So, this is not the thing which you can see here. So, this is where the difference in analysis and the LTSpice simulation will come. You can try this kind of circuit just to understand the operation of the circuit, how it will be in the experiment. Okay? So, let me just check whether the simulation is completed or not.

Refer Slide (43:50)



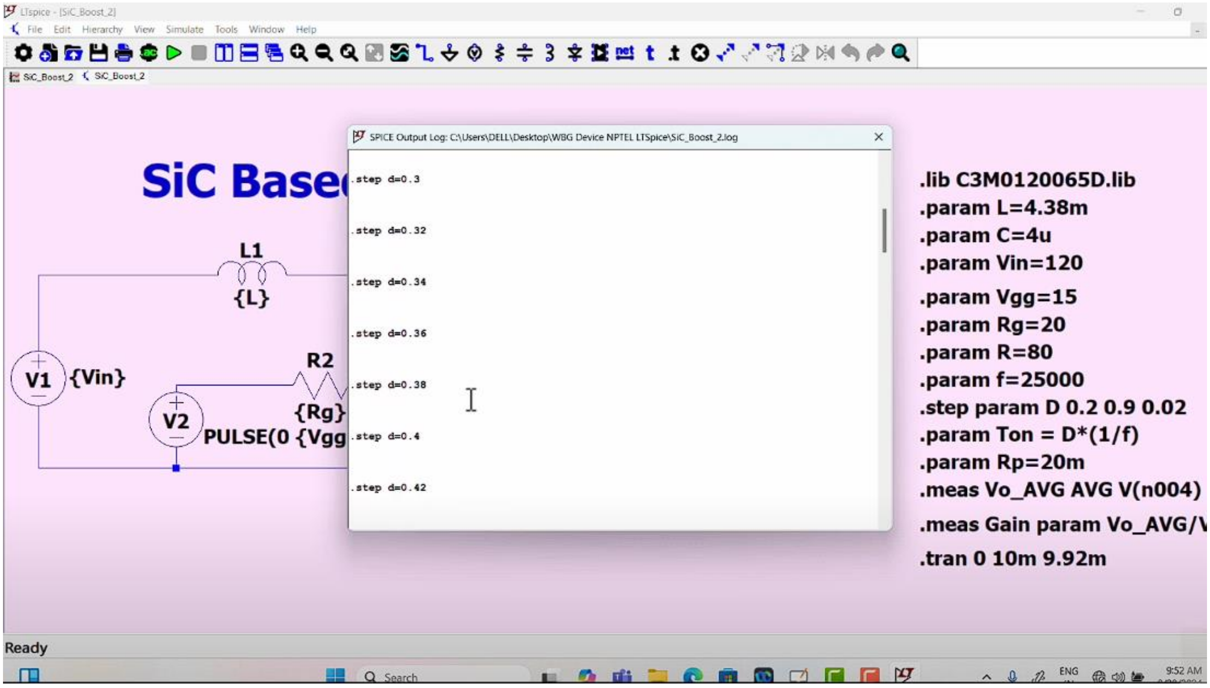
So, now this is completed. Now what we can do? So you can see this different inductor current. Why this different color you can see here? So this is because of duty cycle is varying.

Refer Slide (44:05)



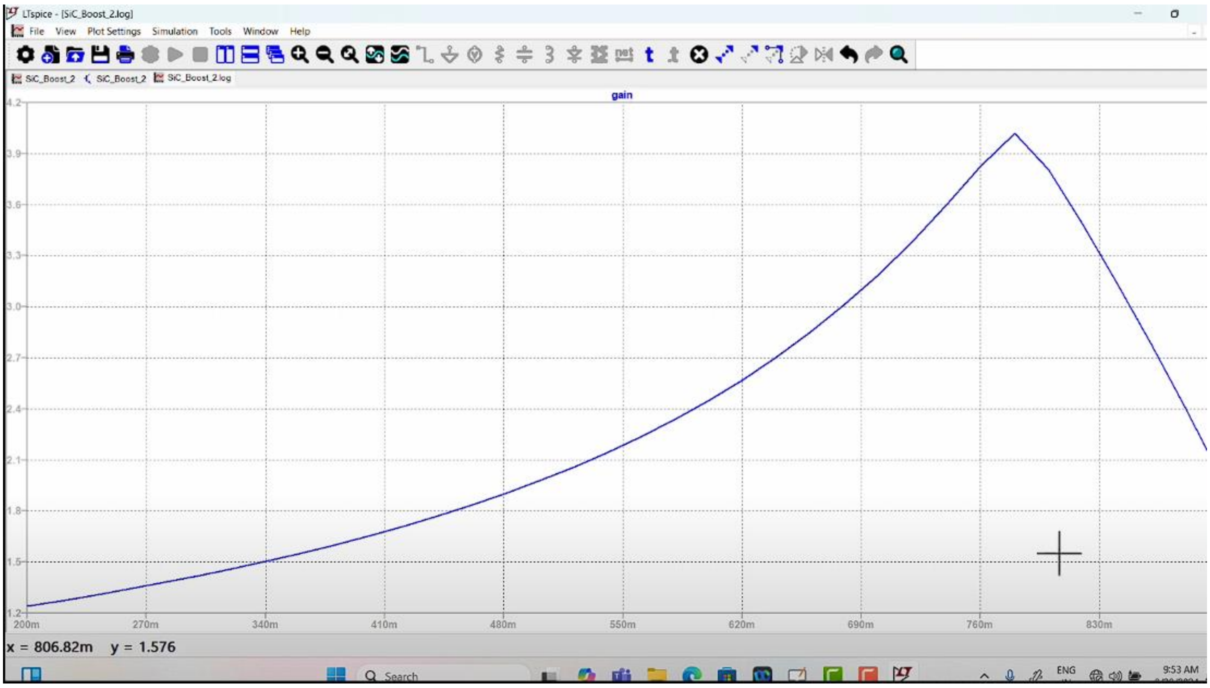
So let me just show you here. So the duty cycle is varying. So based on the duty cycle, this inductor current will also be variable. So when duty cycle is 0.2, so that time the inductor current will charge only for 20% duty cycle. When the duty cycle is 90%, the inductor will charge till 90%. So, this is why the different color in the inductor currents are coming. So, it shows you that it has considered different duty cycle in the simulation itself. So, now we will just show you the gain plot. So, gain plot you can just see. So, you can already average is given.

Refer Slide (44:53)



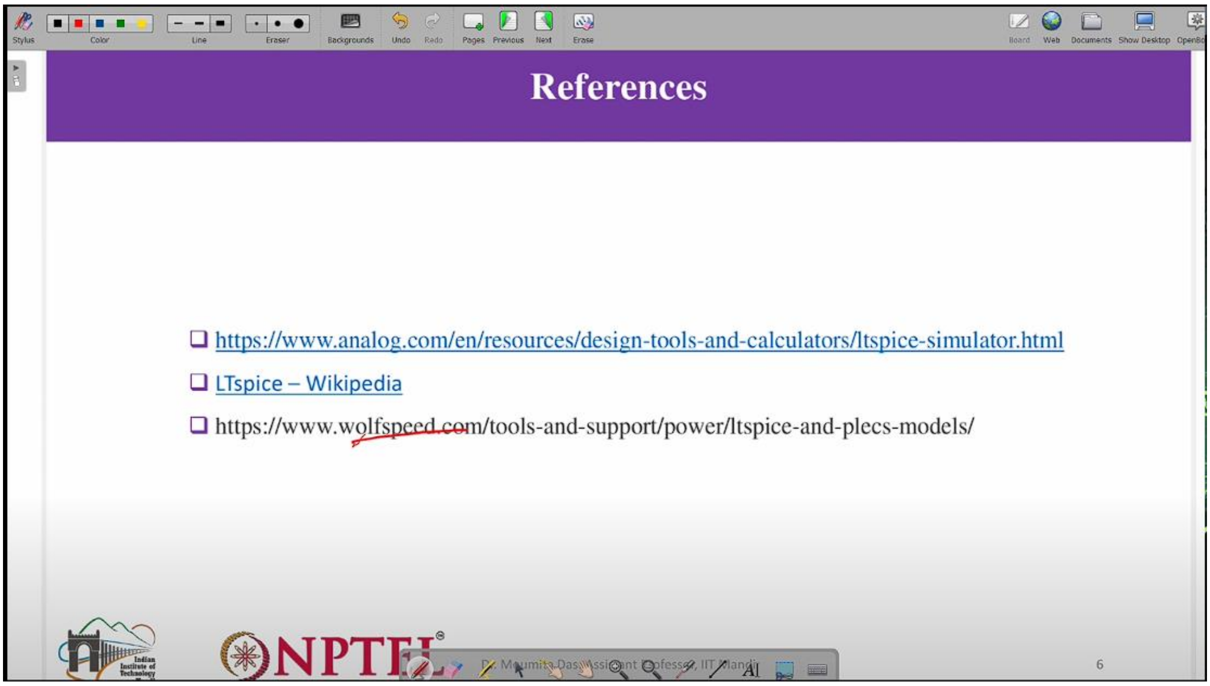
So the steps are given here. These are different steps you can see here. So this is taking in the average. And now you can see here this average. So this step 1 what will be the voltage, average voltage. Step 2 what will be the average voltage. Step like this you will get different average voltage. And then input voltage already you have. So then from there step 1 what will be the gain. Step 2 similarly you can see this measurement of gain you can get. The same data you can just plot in excel sheet.

Refer Slide (45:48)



So then you can or you can just plot here. Plot. So, we are selecting gain versus duty cycle. So, you can see this is the gain versus duty cycle plot you will be getting. So, this is the thing I just wanted to show you because this is something which you cannot get directly from the current and voltage. This is something analysis part is involved. to calculate the gain and to give different steps in duty cycle. So, this kind of thing you can also do in the simulation. Okay? So, now you understand how this practical simulation can be done. So, this is the references of this particular simulation.

Refer Slide (46:50)



You can just go through it to understand more about this like you can just go through this Wikipedia of LTSpice and then downloading part will be done in the analog.com and then the device which we have used this is from the wolfspeed. So, similarly you can use GaN system switch. So GaN system switch is given in their website like wolfspeed.com. So you can go to the GaNsystem.com and then from there you can select any model of the GaN switches. Use the same model in the similar type of boost converter and see whether any difference is coming or not. You will see some differences because you know the parasitic parameter of the GaN and the silicon carbide it will be different. Even if you keep the same condition, some differences will come in the switching or the gain, some small differences you will be able to notice. Just check that to understand basic of this simulation. How to use the wideband gap devices in any kind of converter for simulating the converter. This you can get from these two lectures. Try it by yourself at home. Okay? Thank you.