Control and Tuning Methods in Switched Mode Power Converters

Prof. Santanu Kapat
Department of Electrical Engineering
Indian Institute of Technology, Kharagpur

Module - 02
Modulation Techniques in SMPCs
Lecture - 12
Interactive MATLAB Simulation and Case Studies

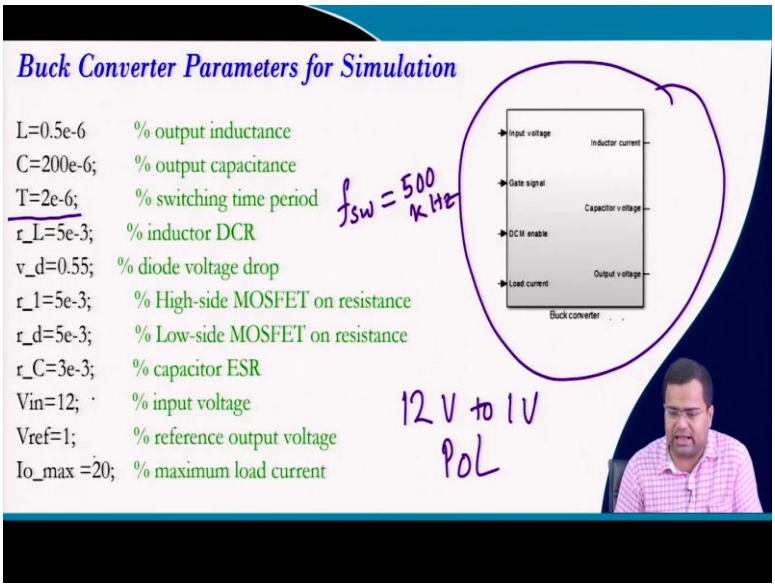
Welcome back. Today is our 12th lecture and, in this lecture, actually in the previous lecture, we developed Simulink model for buck as well as boost converter buck converter where we have also shown how to incorporate discontinuous conduction mode. So, now, we are assuming that using the same method, you can build such buck and boost converter Simulink model, but now, we want to create multiple test cases.

(Refer Slide Time: 00:59)

So, in this lecture, I want to show like a instead of because such test cases will be very difficult to execute you know by running in Simulink, but we want to make such interactive case study from the MATLAB script file itself. So, we want to run Simulink and using script file, we want to process data and we want to create multiple transient scenario you know customize them, then we can you know various thing we can do more interactive.

In fact, we can speed up the simulation by running through script file. So, Simulink and script file interactive simulation, then creation of various transient test cases, then fixed frequency and variable frequency modulation I want to show in this lecture.

(Refer Slide Time: 01:50)

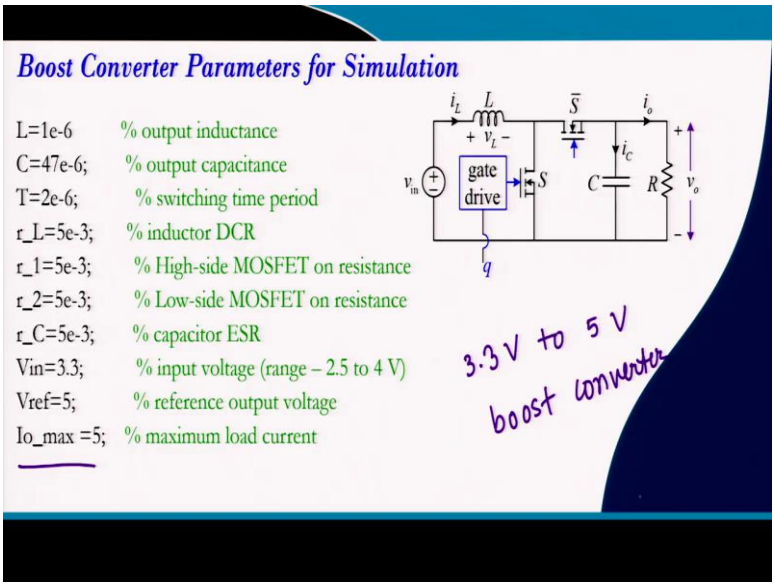


So, this block I am showing, this particular block actually, we have shown this block in the previous lecture that mean this how to build this buck converter block and we will use that block to move further. For today's lecture, we have taken a separate inductor resistance value and I will discuss that how did we arrive such parameter values like inductor, capacitor value and here, we are talking about a switching frequency of f sw target frequency of 500 kilo Hertz.

So, that is why time is 2 microseconds. So, in the subsequent lecture, may be the next lecture in the 3rd week. We summarize whatever we have derived various rms current, ripple current, output voltage ripple. So, using those ripple criteria, how to select inductor capacitor value for a given switching frequency.

So, those part we will discuss, but today we are taking it for granted that the inductor, capacitor, these things are given and we are talking about some 12 to 1 volt so, that means, we are talking about 12 volt to 1 volt POL which is the buck converter that we are going to consider in today class.

(Refer Slide Time: 03:29)

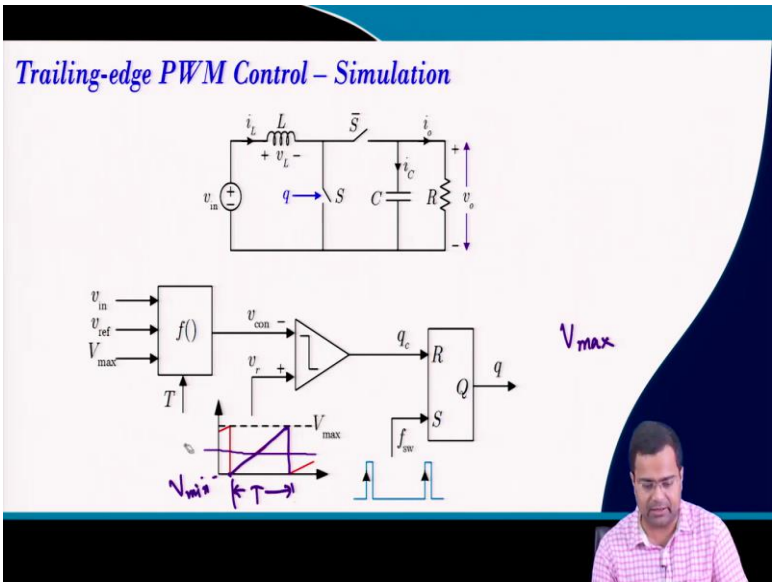


We will also consider a boost converter where the inductor is 1 micro Henry, the capacitor is 47 microfarad. We will discuss this selection of inductor capacitor in the next lecture and here, we are talking about a 3.3 volt to 5 volt which is the boost converter simply boost converter because we are stepping up the output-input voltage. So, output voltage is higher than the input voltage and we need to define here what is the maximum load current? I0 max.

But if you look at two cases, in the first case, we are talking about a conventional buck converter and in the Simulink, we can actually use a synchronous buck by changing the DCM unable logic and we also want to change the diode drop as well. Because in synchronous buck, there is no diode and there is no question of diode drop.

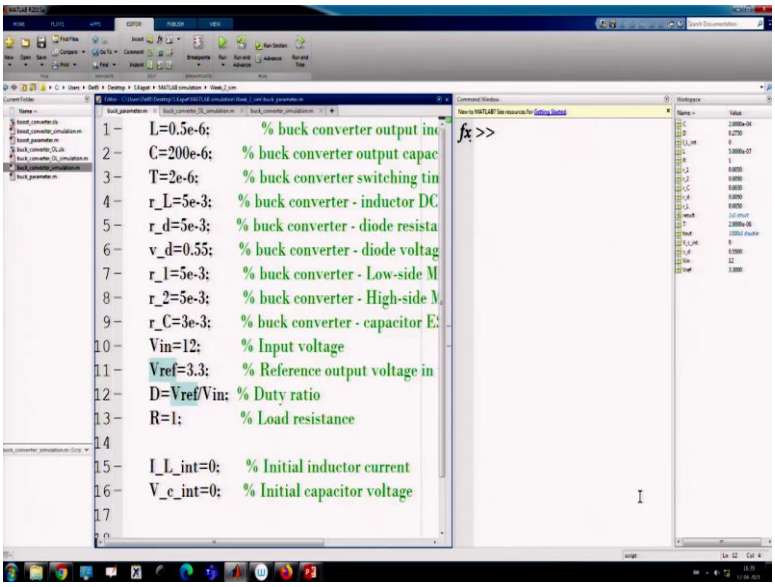
In boost converter, we are talking about a synchronous boost, but you can also take a conventional boost, it does not matter. Why we are talking about synchronous boost? Because we want to see the dynamics of boost converter in continuous conduction mode, and we want to see what happen you know what is the effect of right up and 0 and so on.

(Refer Slide Time: 05:02)



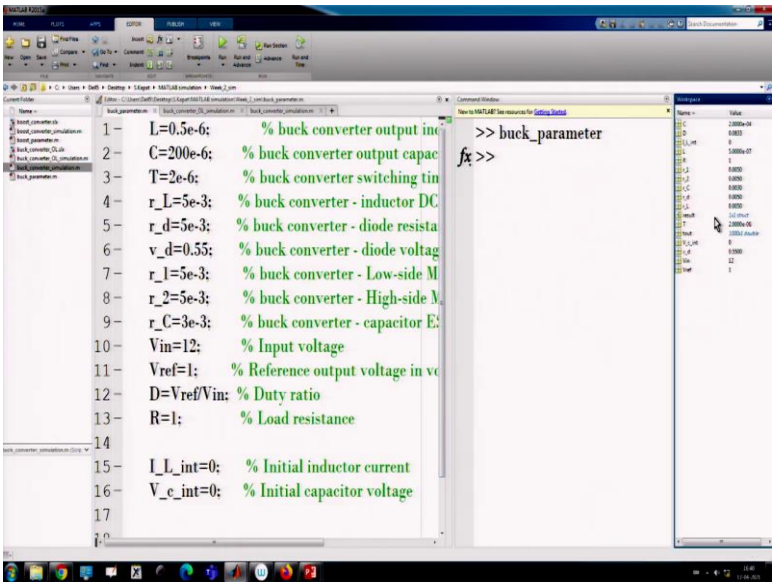
So, with this, we will start our simulation, first we will run the open loop simulation ok.

(Refer Slide Time: 05:18)



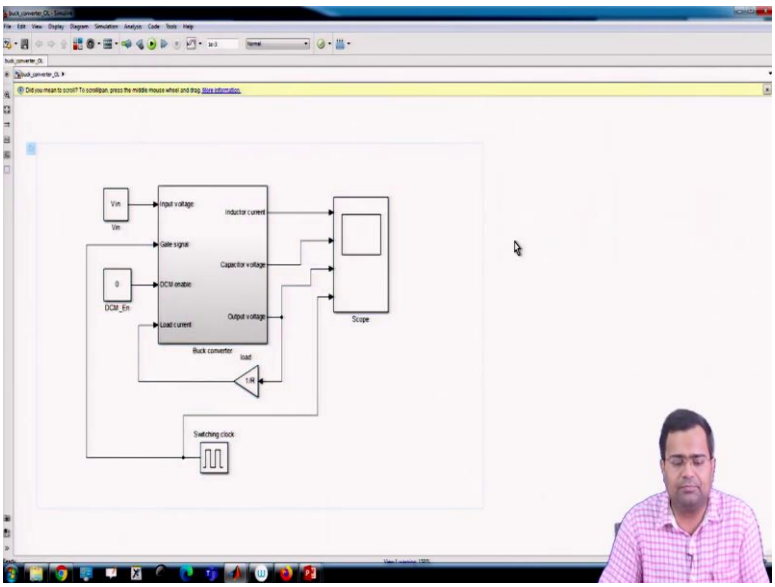
So, let us go to the open loop simulation. So, in this simulation, we have considered this parameter which I have shown 0.5 inductor, 0.5 micro Farad, capacitor is 200 micro Farad. The time period is 2 microsecond, r L that is the DCR of the inductor, then diode resistance, diode voltage drop, then MOSFET on resistance, MOSFET of the low side we are not using so, this is still there, 3 micro; 3 milliohm resistance for the ESR, 12 volt input.

(Refer Slide Time: 05:59)



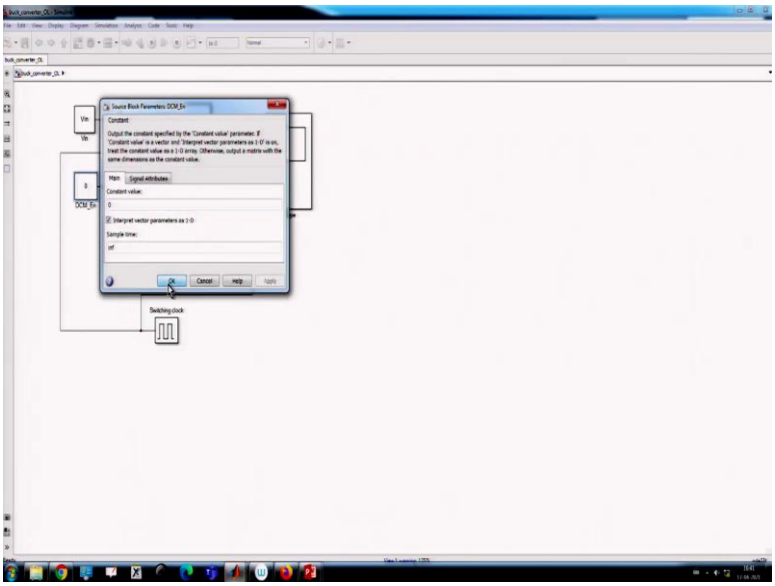
And 3 point so, here we are talking about 1 volt output not 3.3, 1 volt output and load current can vary. So, these are like you know we need to define all these in the custom Simulink file, all these initial conditions everything, we will create a test case. So, first, if we run this, we know that the parameters are executed, and these are all stored in this workspace. So, you can see this workspace.

(Refer Slide Time: 06:23)



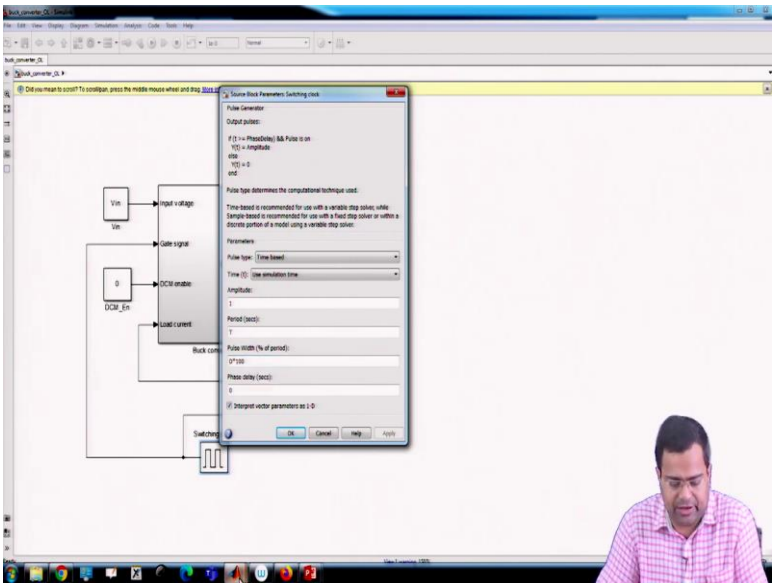
Now, we are going to Simulink file. So, in this Simulink file, we have said DCM enable logic to be 0.

(Refer Slide Time: 06:29)



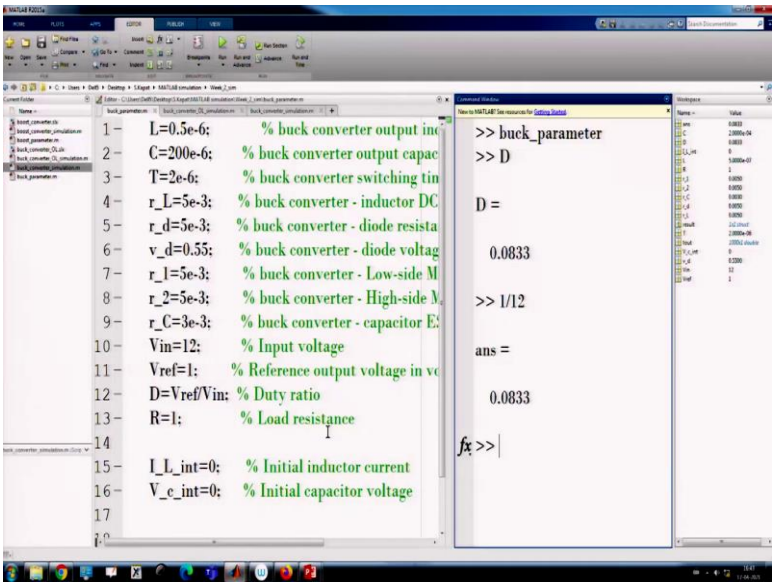
That means, we are not going to talk about; it is just we are trying to operate in continuous conduction mode ah; we are trying to operate in continuous conduction mode. This load resistance here we have set as 1 ohm.

(Refer Slide Time: 07:00)



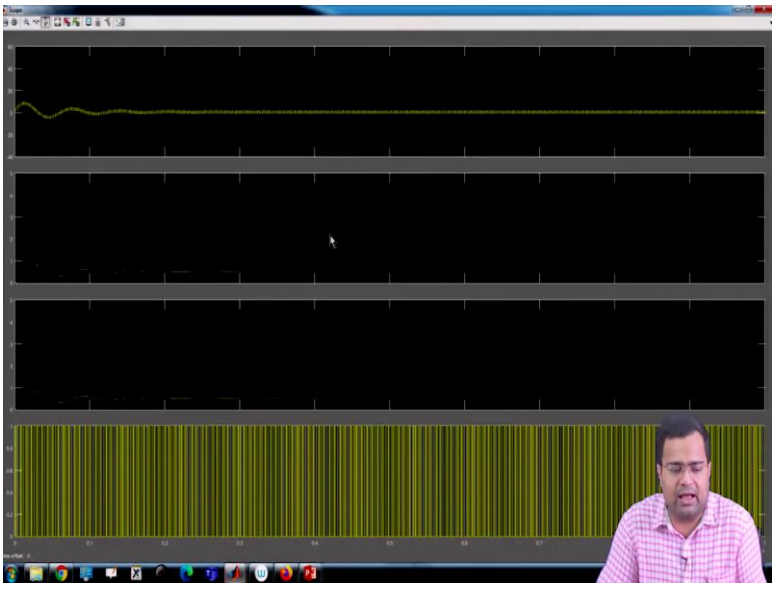
I have taken a clock or time period T and which is nothing, but our switching time period of 2 microsecond and duty ratio D, I said which is Vref.

(Refer Slide Time: 07:17)



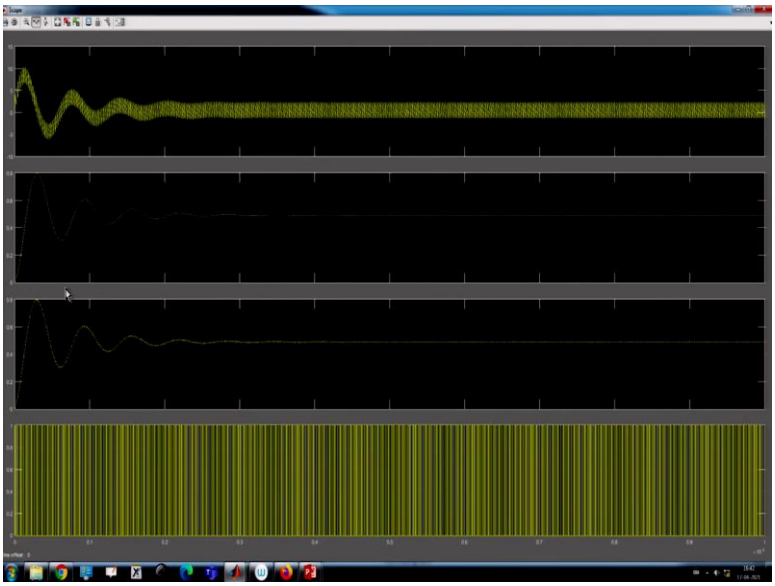
So, if you execute this comment, what is D? You can check D we got 0.0833 because we are talking about 1 by 12 volt, this value. Next DCM level 0, input voltage 12 volt, load resistance 1 now, we want to run the simulation.

(Refer Slide Time: 07:39)



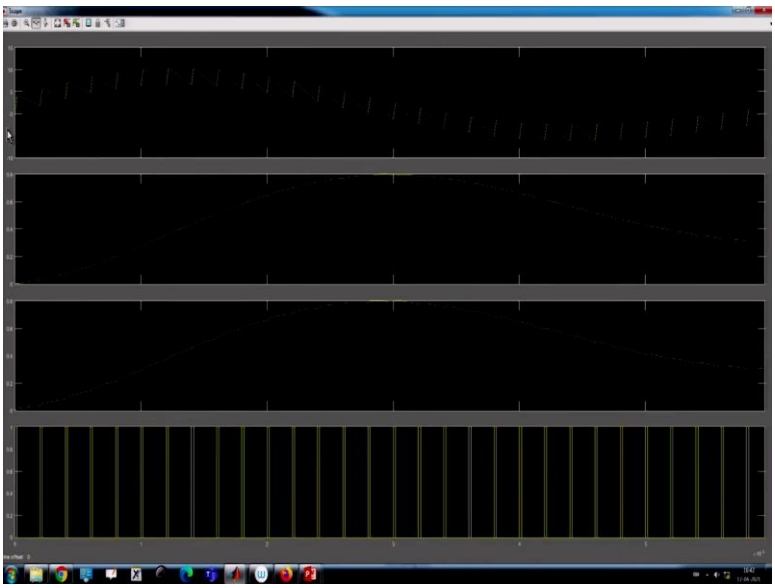
If we run the simulation so, this is starting from 0, all initial condition initially was 0.

(Refer Slide Time: 07:42)



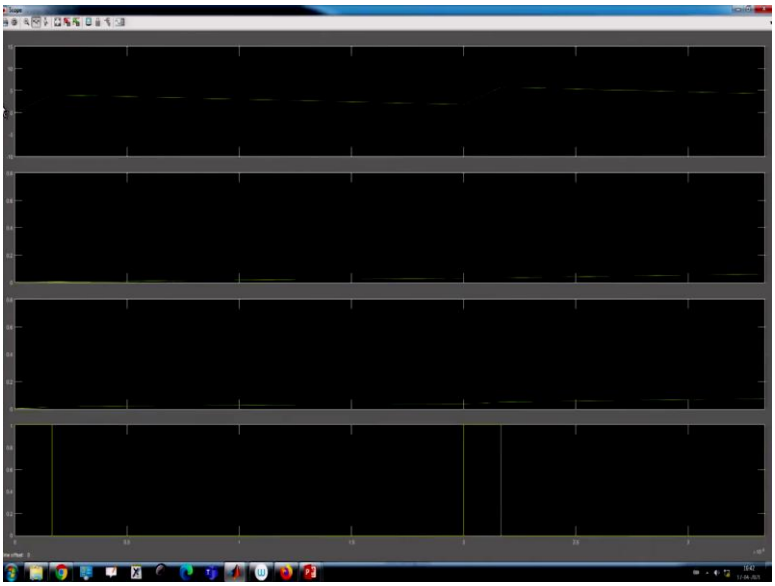
So, inductor current start from 0, output voltage start from 0. So, what we are showing here?

(Refer Slide Time: 07:47)

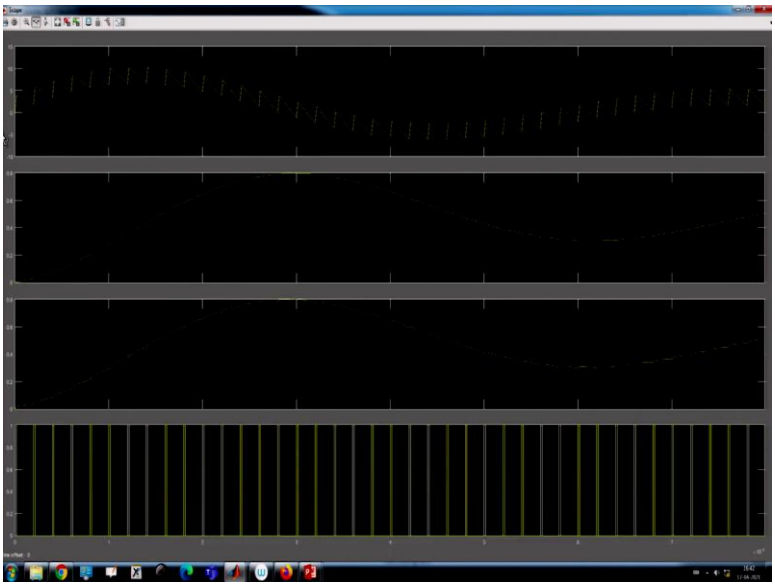


The first scope is our inductor current.

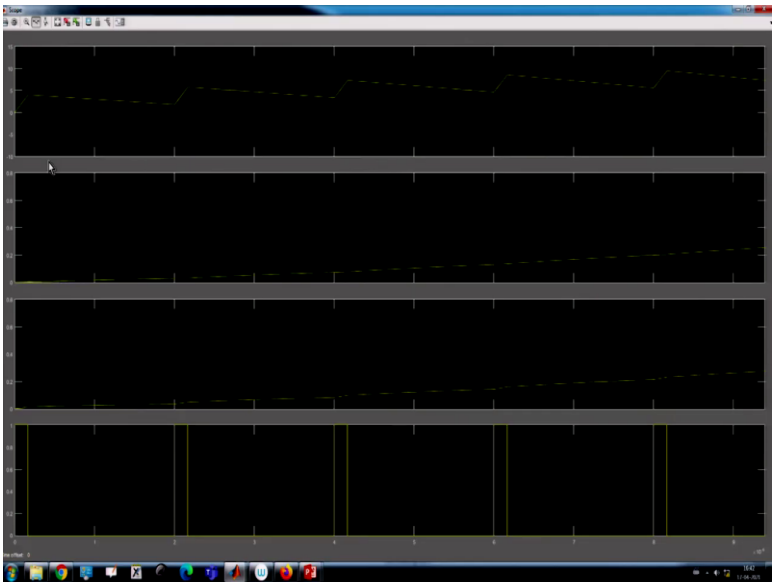
(Refer Slide Time: 07:50)



(Refer Slide Time: 07:57)

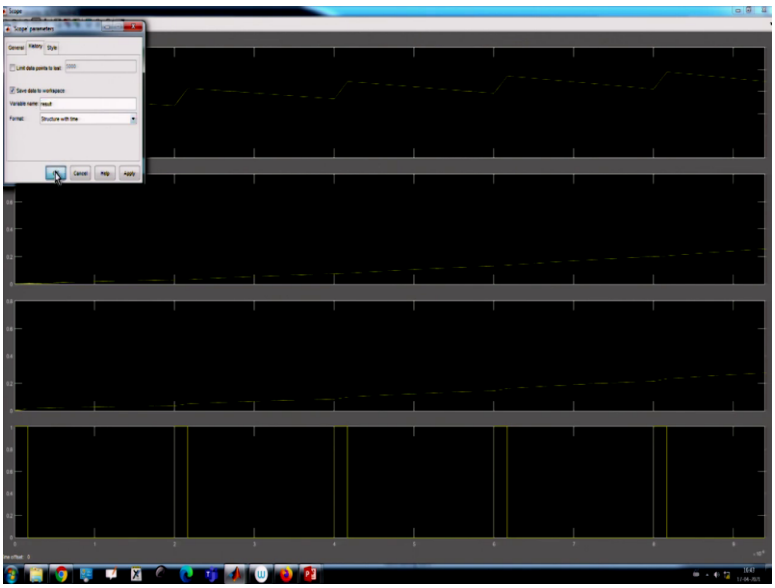


(Refer Slide Time: 08:02)



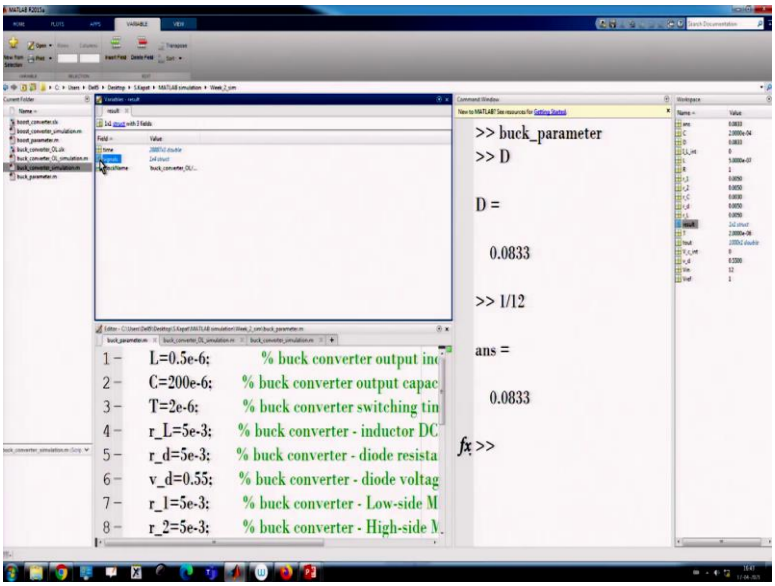
The second scope is our capacitor voltage. If we look at what are there, second is the capacitor voltage, third is the output voltage and the fourth is the our switching clock ok.

(Refer Slide Time: 08:19)



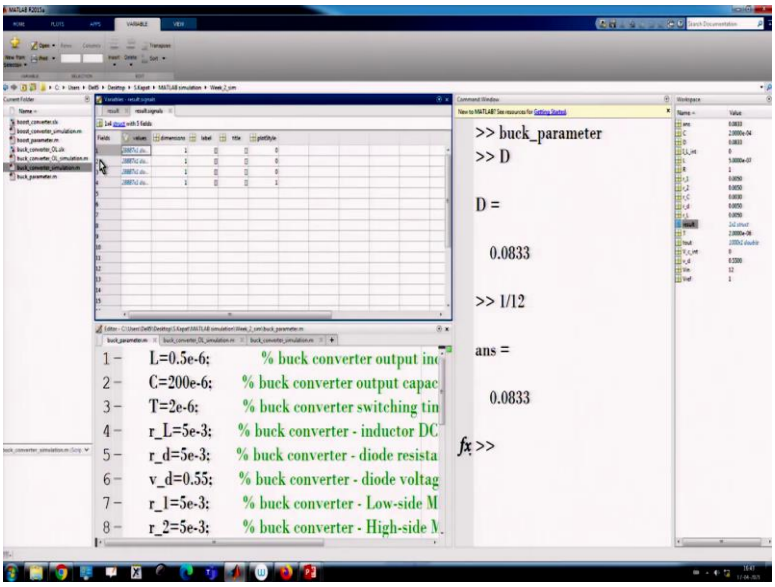
And we also saw in the previous class that if we want to store this data into workspace, then we have stored in the name of a result which is a structure and we can see that structure in the workspace that is the structure.

(Refer Slide Time: 08:33)



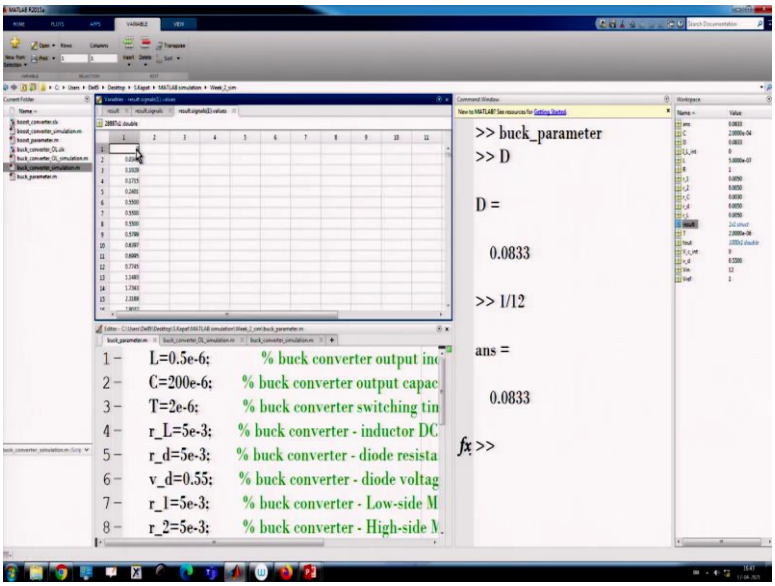
And which is the time vector as well as signal ok.

(Refer Slide Time: 08:37)



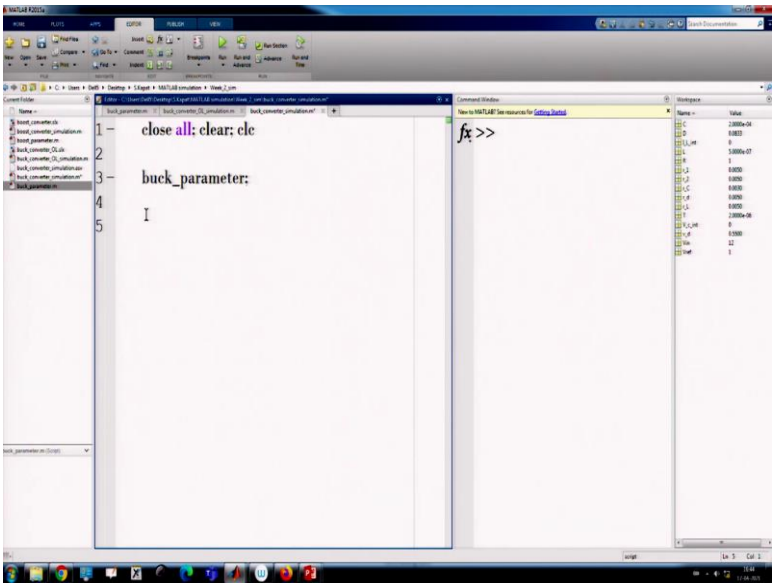
So, we have 4 channels are there. So, each channel, if you click, then you will get one by one data.

(Refer Slide Time: 08:41)



Now, we want to create a text file because instead of running the Simulink every time; we have a parameter file and we can, in fact, customize some of the parameter during the runtime that is also possible.

(Refer Slide Time: 09:05)

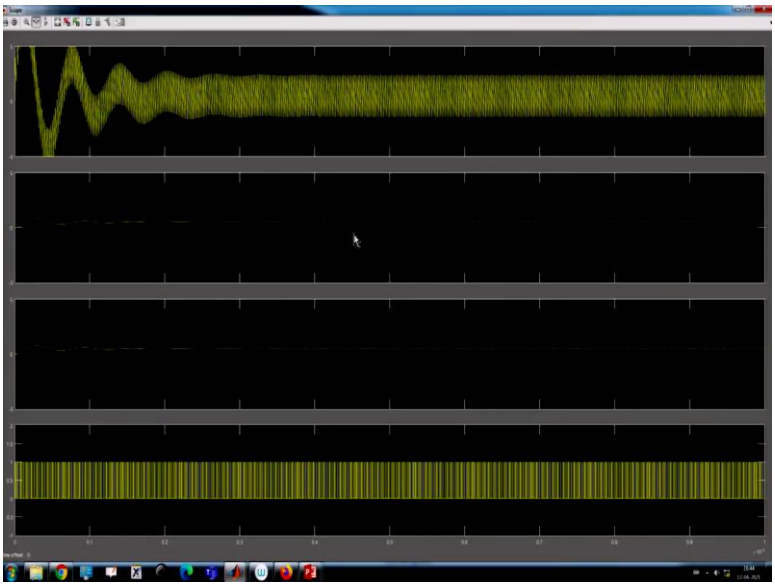


So, here what we will do? First, we will clear the screen close all so, this will close all these in a workspace sorry all the figures and everything, then clear, it will clear all the workspace, and this will clear the comment from. So, if we execute these three, then what will happen? Everything is now clear.

Now, we want to call the simulation, the parameter file because if you want to run this through simulation, this dot m file, we need to call this parameter file and this can be simply called just by the filename. If you copy this, you can simply call this filename that is it. Once you run it, then you can see all the parameters are again loaded. So, that means, we are executing this parameter file so, as a result, all parameters are stored.

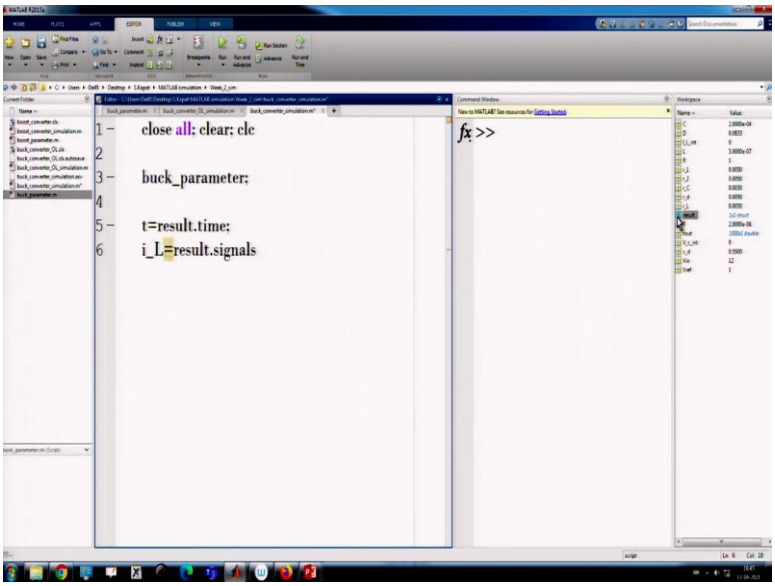
Next, what is our next task? So, after clearing this, we have executed this parameter file now, you can customize the initial condition everything separately here that will do, but before that, we want to first see the scope result. So, let us run again this simulation because we have cleared everything.

(Refer Slide Time: 10:23)



So, now, it is running.

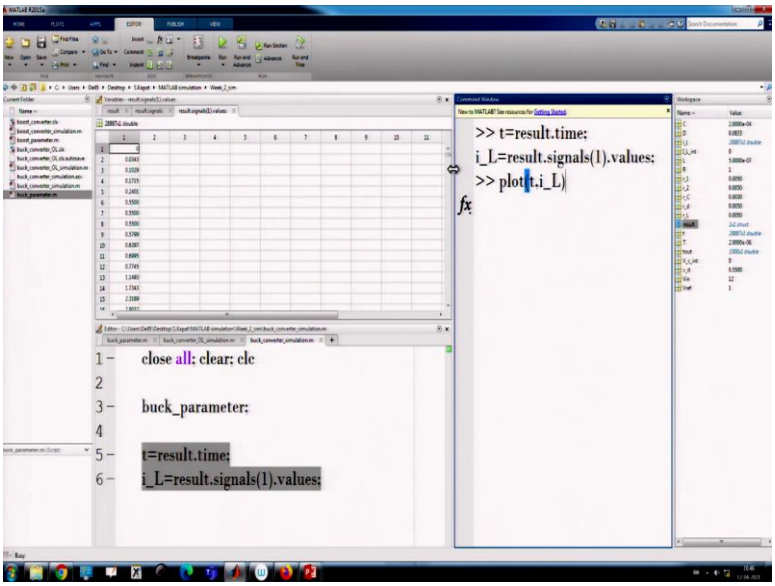
(Refer Slide Time: 10:30)



Now, we want to take this data into our it is there in the workspace so, we want to create t equal to result that is our vector that we have discussed that in the structure comprises one time vector, the other signals structure again its signal structure have different values like different channel. Then, we want to see what is the inductor current? So, inductor current is our it is in which channel?

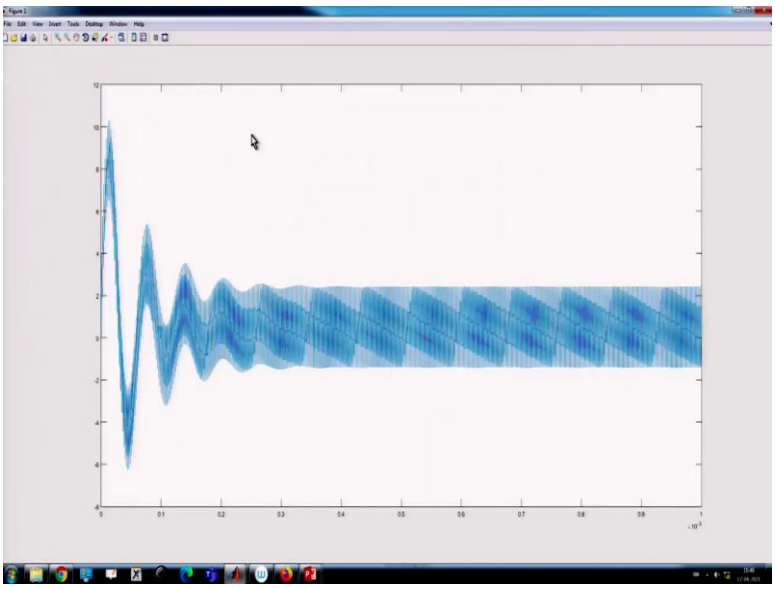
Result dot signals because if you click the result, you will see the signals and inside signal if we go, the first channel if we look at this, the first channel inductor current, then capacitor voltage, then output voltage, then the gate signal ok. So, let us go back.

(Refer Slide Time: 11:29)



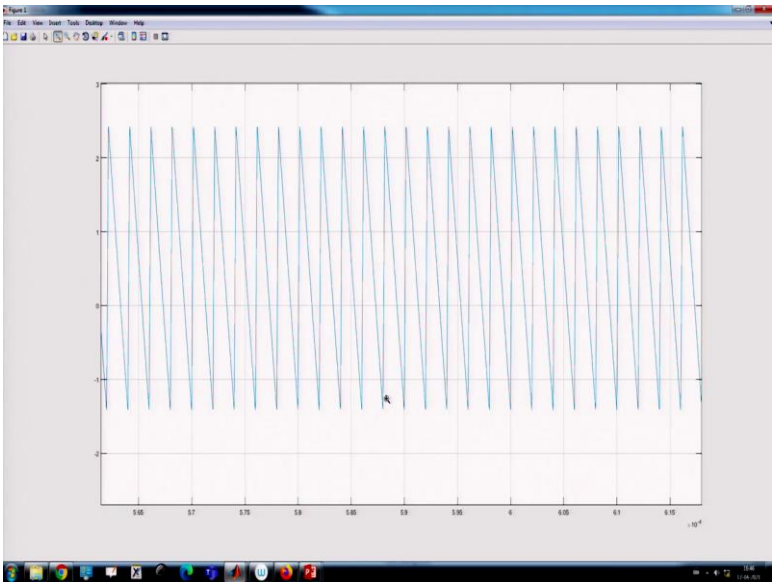
The first is our inductor current and if you look at this one, this result dot signal 1 dot values so, this is what we have to type signals 1 dot values ok. If we execute these two, just these two particular lines, then we can plot, we can plot t comma i L mean inductor current we can plot.

(Refer Slide Time: 11:48)



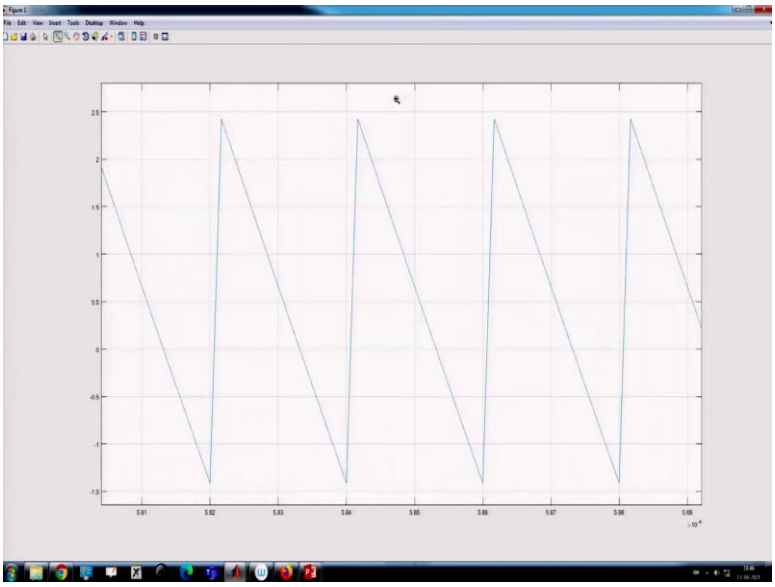
Yeah, this is the plot. So, now, we can plot, we can actually create a grid here.

(Refer Slide Time: 12:01)

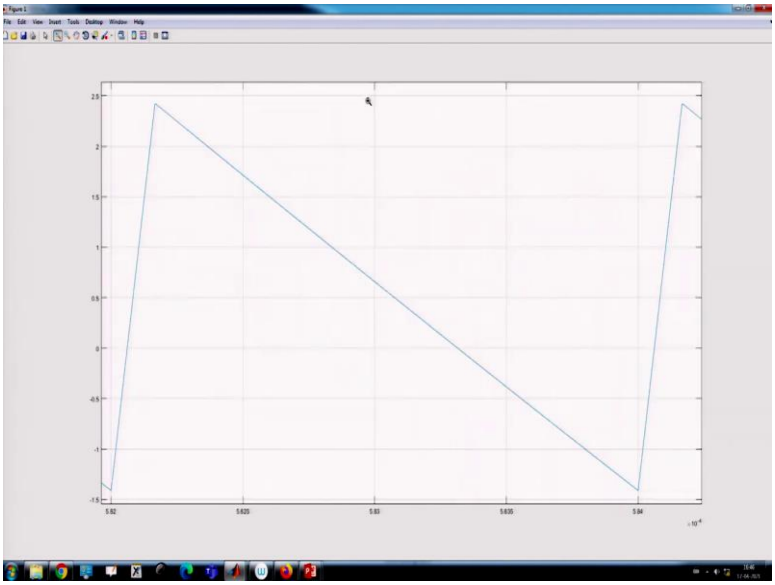


And we can see what are the current waveform, the inductor current waveform, so on.

(Refer Slide Time: 12:03)

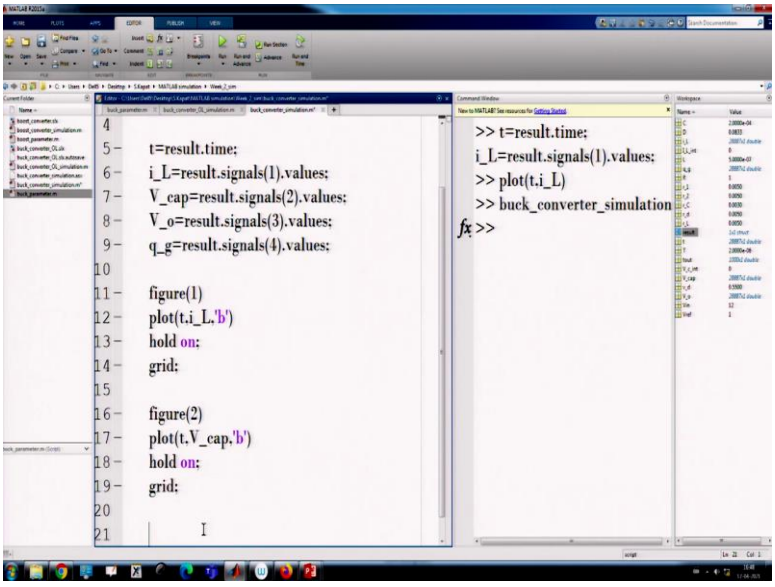


(Refer Slide Time: 12:06)



So, this is our inductor current waveform of a, buck converter because the duty ratio is very low ok. So, you can call this waveform.

(Refer Slide Time: 12:17)



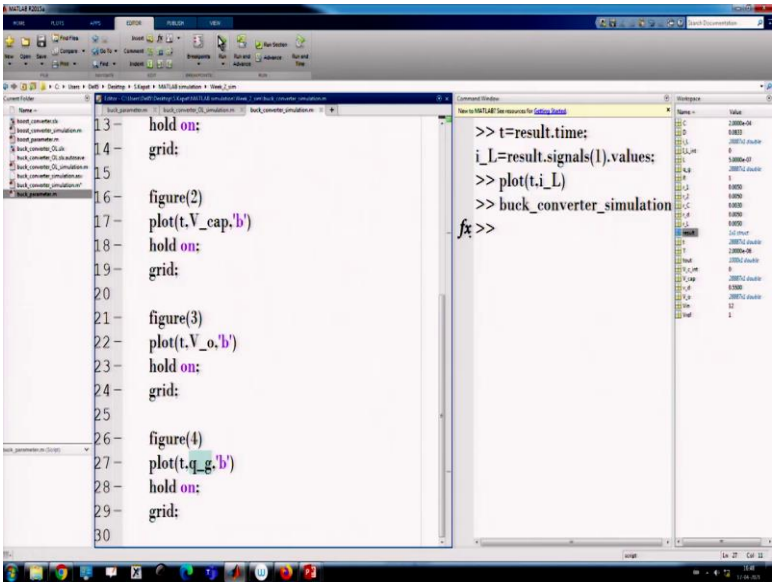
So, by that way, we can call like V cap V capacitor voltage again. We have to only change this channel, everything is same, the 2nd channel which is 2, number 2, then we want to show output voltage it is in the 3rd channel 3rd channel and then, we want to see gate signal q g; q gate q g, it is in the 4th channel.

So, we got now, if we execute, we can get all the data, but since you know if we close the whole thing, then it will actually close all the data. So, we will not execute this for

the time being because if we execute so, this data's are now stored now, we can use plot command.

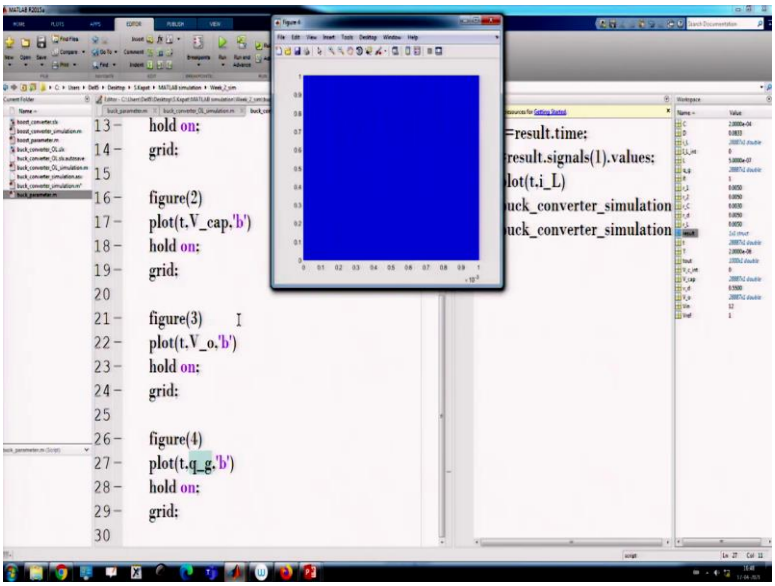
So, if you want to create multiple figures, though you can name like a figure 1, figure 1 for inductor current, we can use plot t comma i L, we can use a difference color. Specific color you can use, then you can hold on, we can grid it and so on, then we can copy and paste for the other waveform. Figure 2, we can draw capacitor voltage, which is our V Cap.

(Refer Slide Time: 13:50)

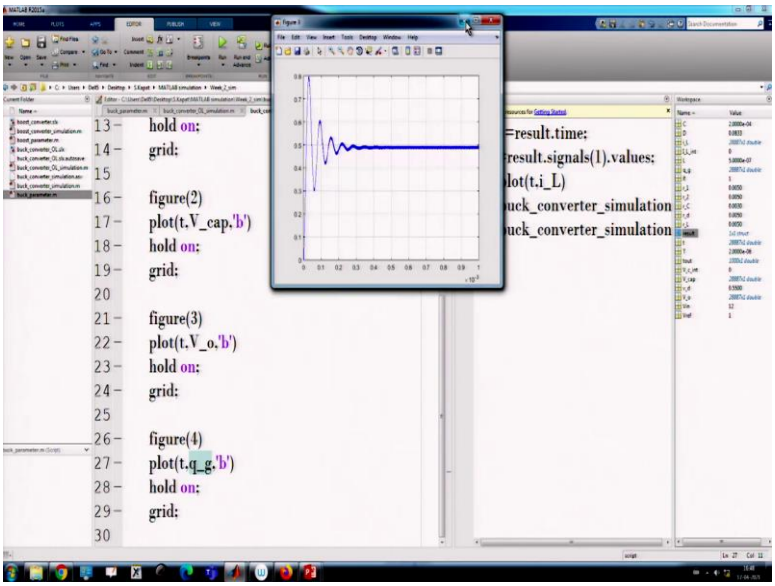


Then, we can make figure 3 which will be my output voltage and then, we can plot figure 4 which is my gate signal q g that is it.

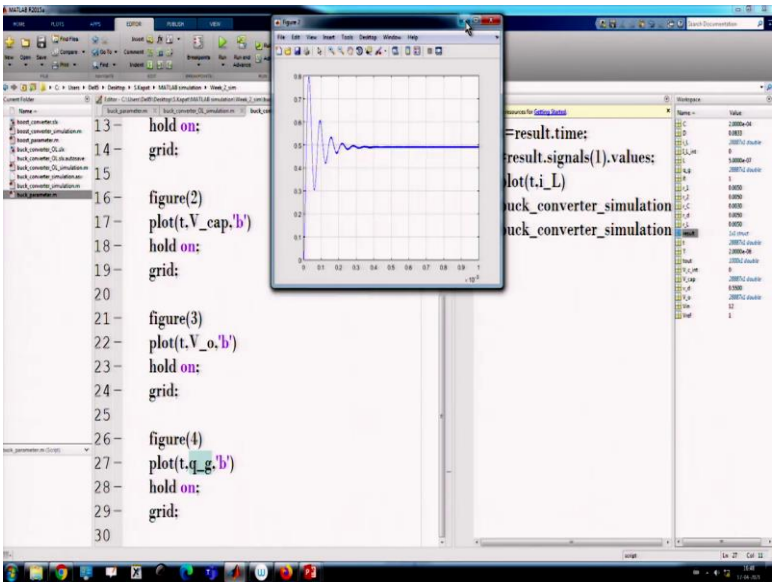
(Refer Slide Time: 14:11)



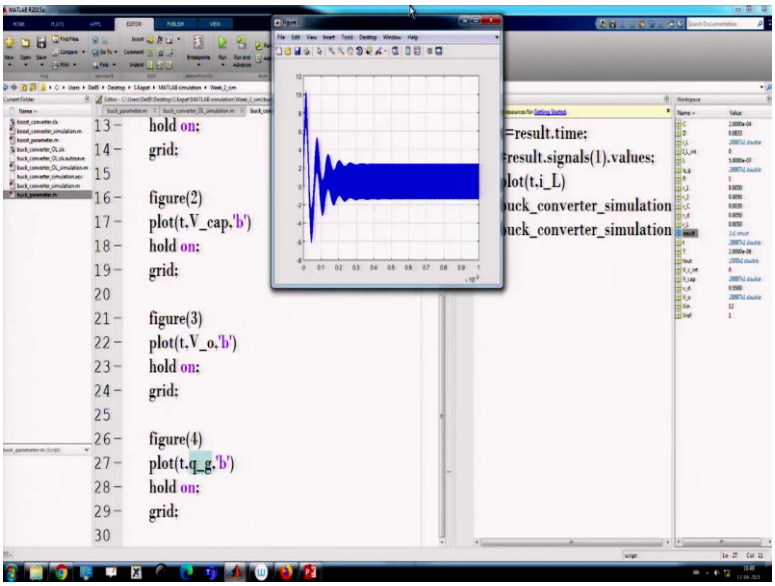
(Refer Slide Time: 14:13)



(Refer Slide Time: 14:14)

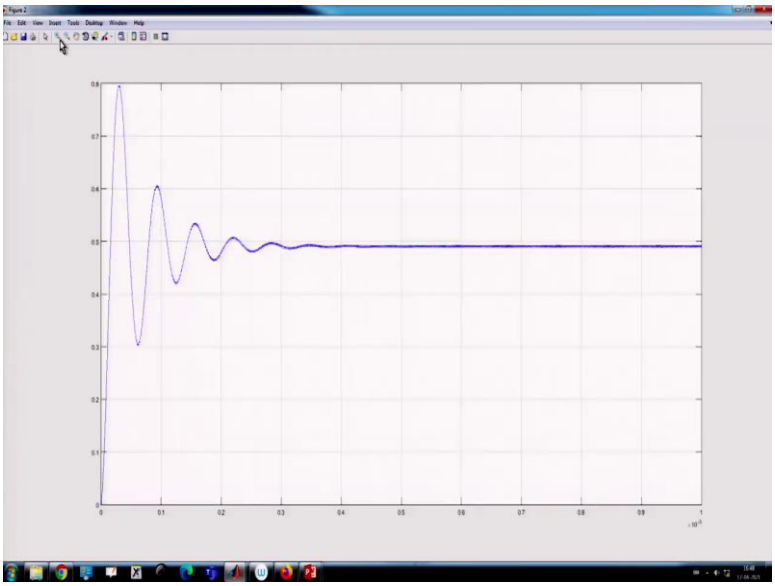


(Refer Slide Time: 14:15)

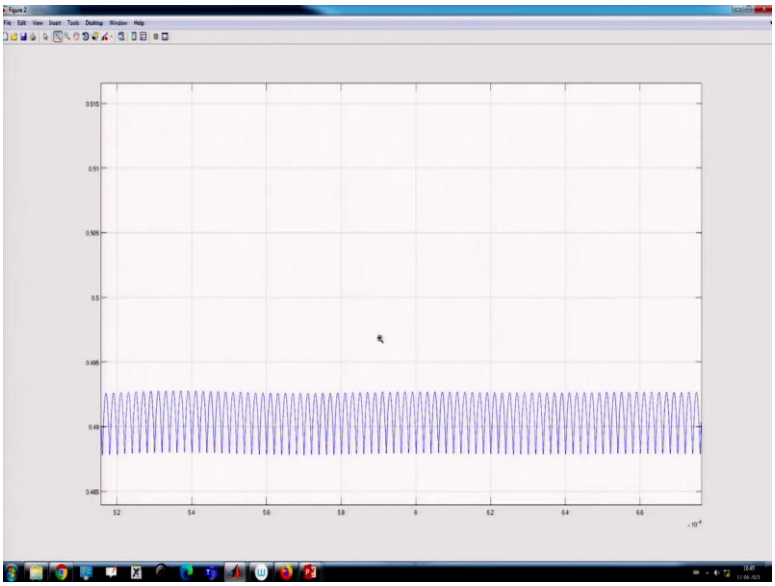


Now, all these four channels can be, you can see all the four channels here right. So, this is my inductor current which I have already shown.

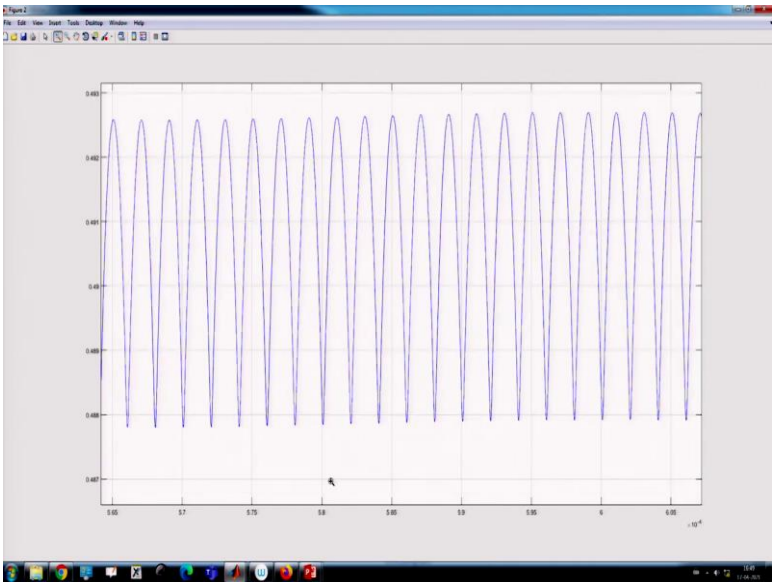
(Refer Slide Time: 14:26)



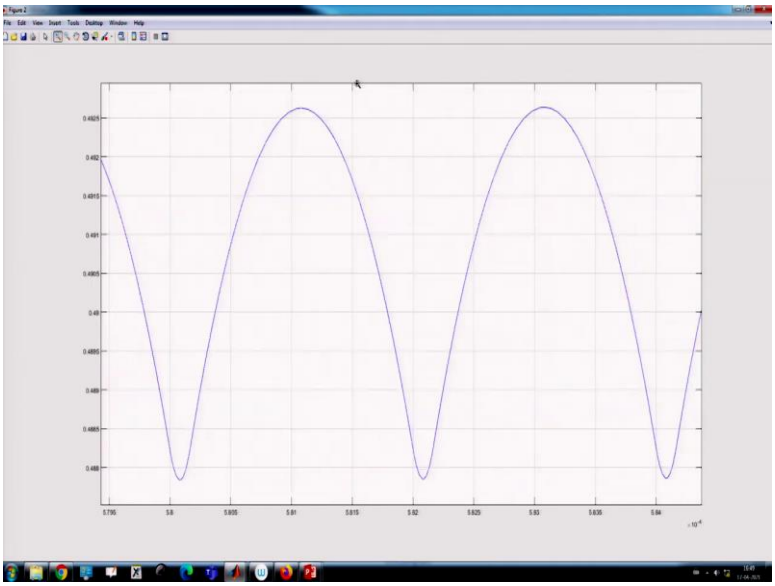
(Refer Slide Time: 14:28)



(Refer Slide Time: 14:30)

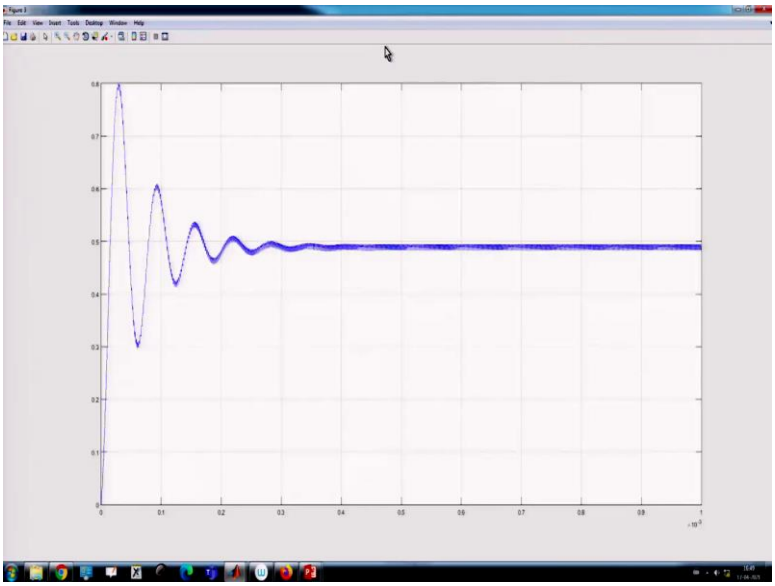


(Refer Slide Time: 14:32)



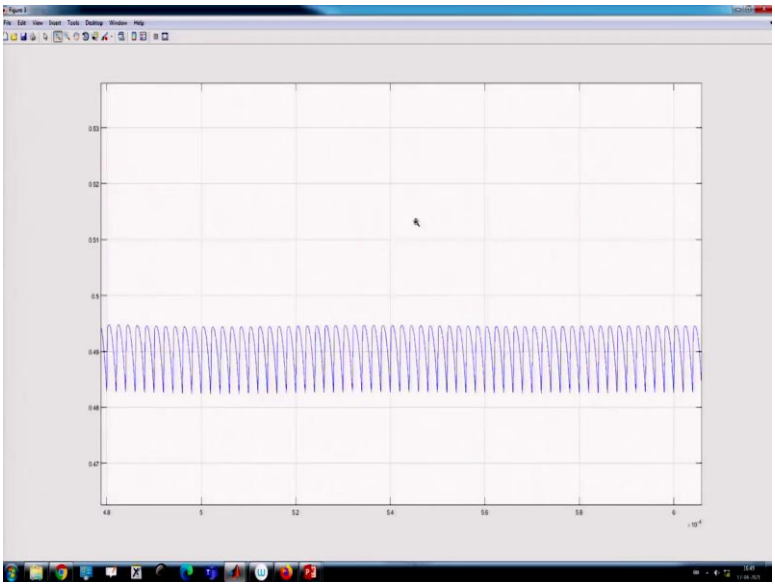
Then, you can see this output voltage, this is the capacitor voltage and if you notice the capacitor voltage ripple looks like this capacitor voltage.

(Refer Slide Time: 14:40)

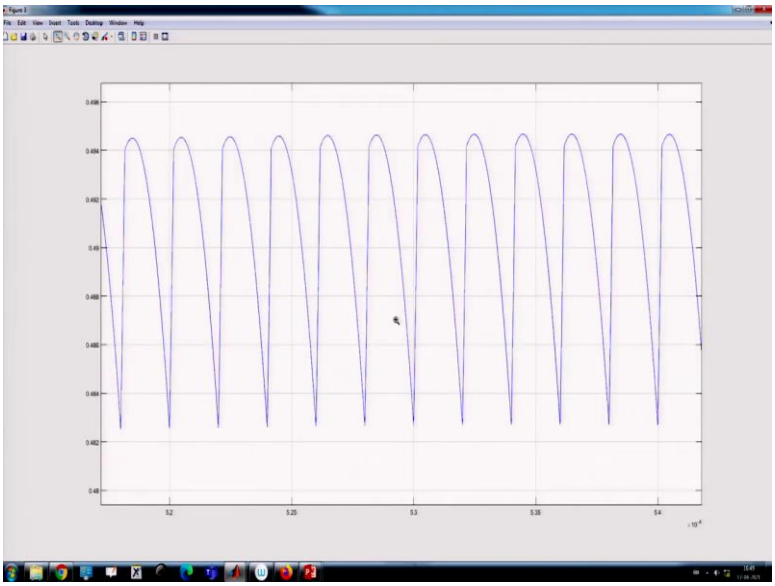


And then, the other channel is the 3rd channel is the output voltage.

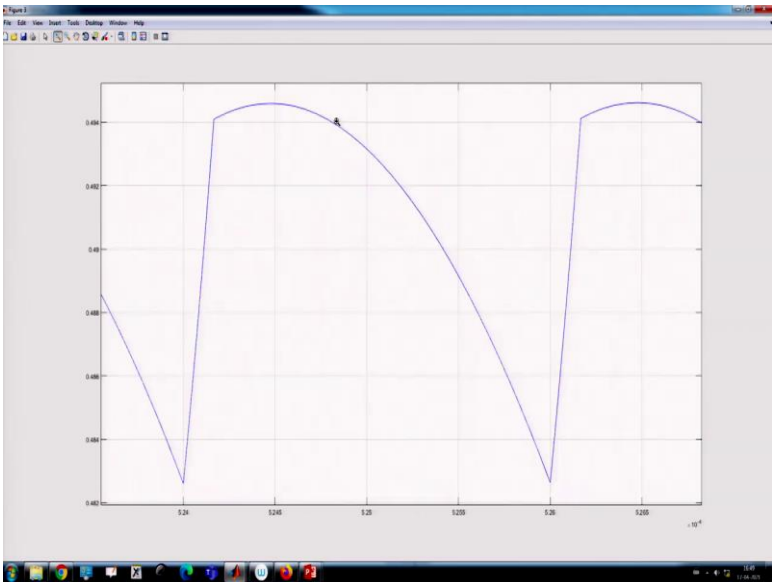
(Refer Slide Time: 14:43)



(Refer Slide Time: 14:45)

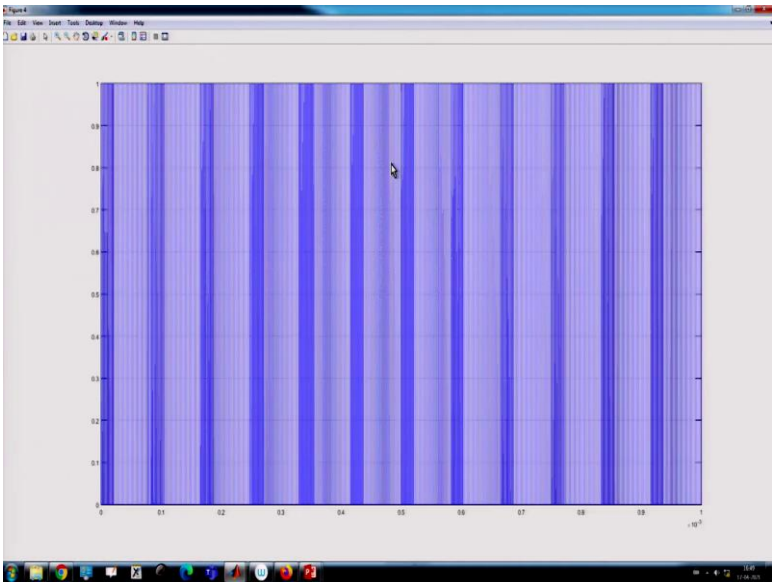


(Refer Slide Time: 14:47)



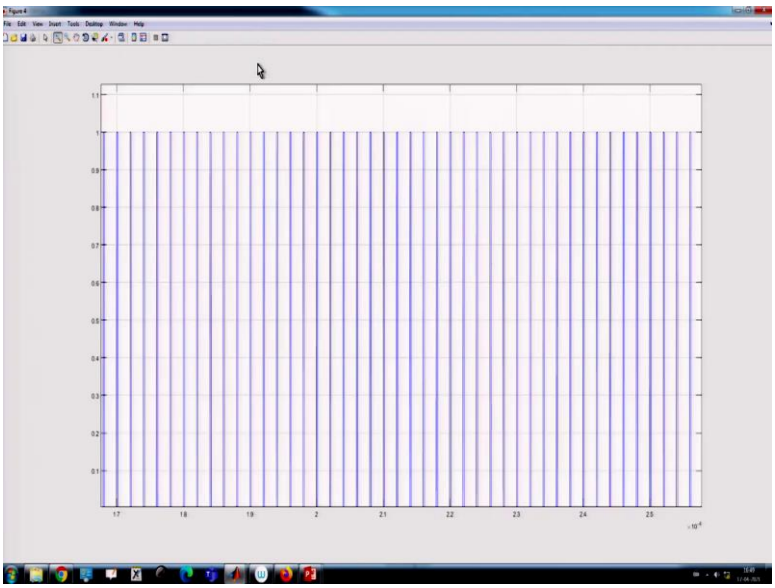
So, this is my output voltage ok, output voltage.

(Refer Slide Time: 14:55)

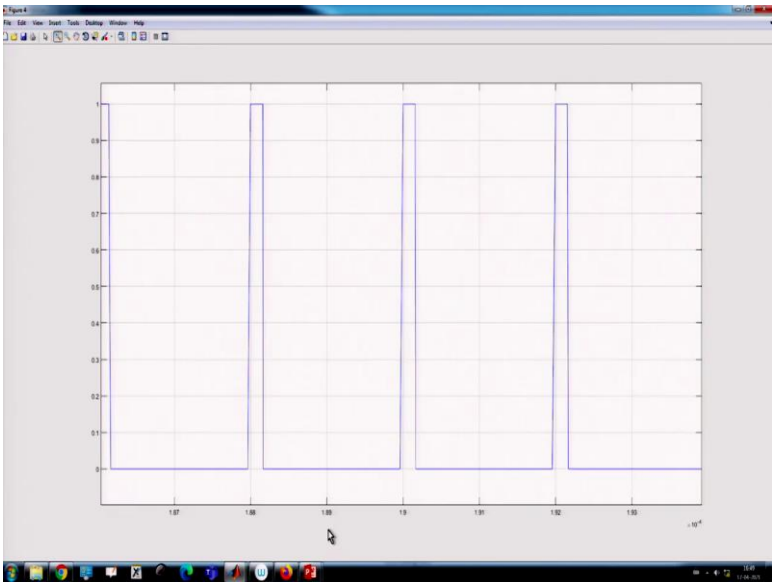


And then the other one and the 4th one is the gate signal. So, this is my gate signal, I can just take a portion of this and just show.

(Refer Slide Time: 15:03)

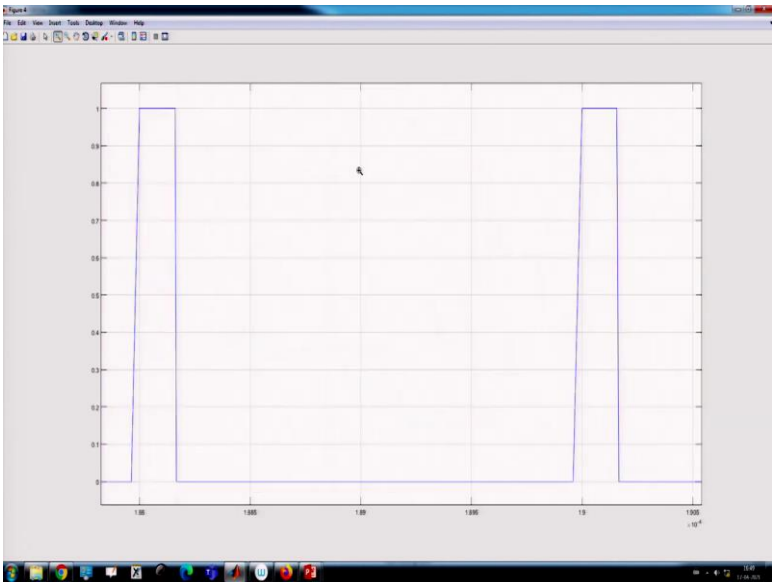


(Refer Slide Time: 15:05)



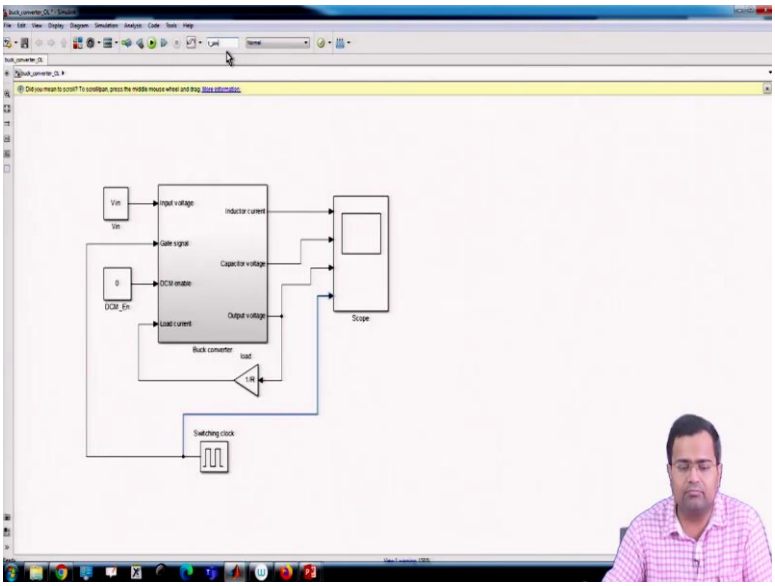
Yeah, this is my gate signal.

(Refer Slide Time: 15:08)



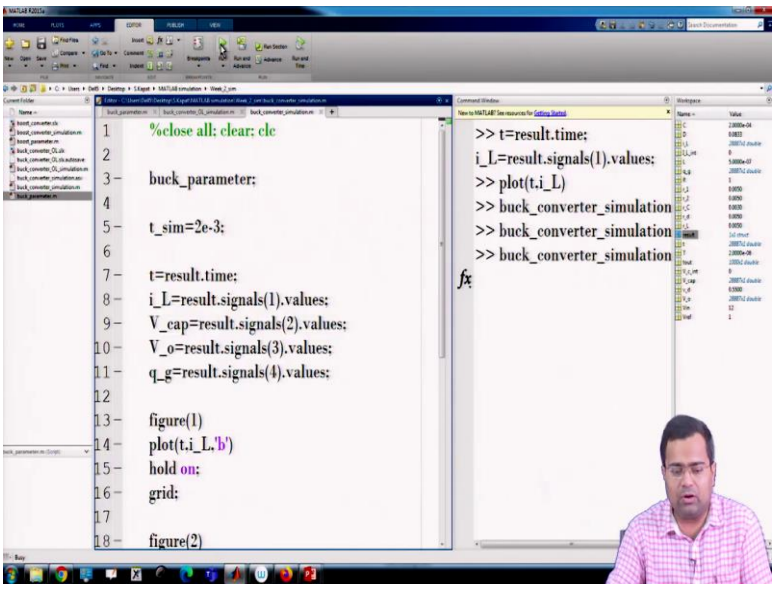
And if you see that a gate signal because the pulse width is very low, it is 12 volt to 1 volt so, its a low duty ratio surface. So, these things are now possible that we can draw it.

(Refer Slide Time: 15:27)



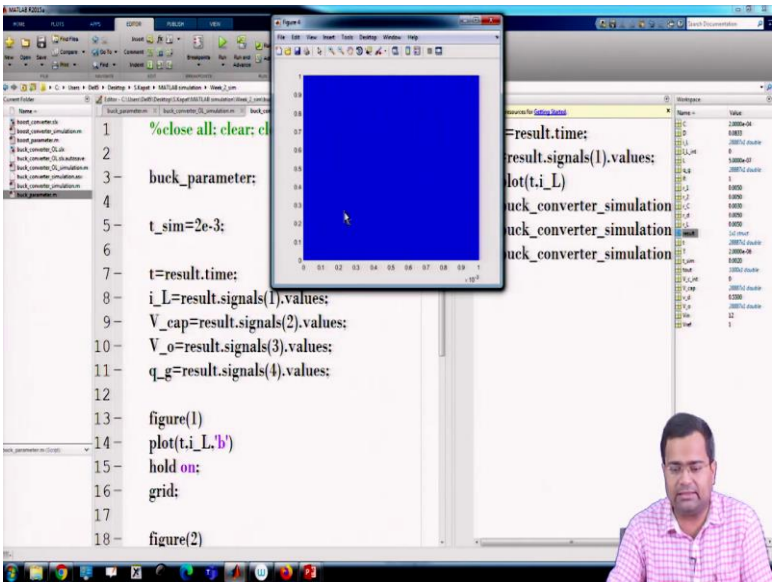
Now, after doing all these, the next question we want to create a transient case that means, we want to change the simulation time instead of 1 millisecond we want to run it 2 millisecond like this so, we have to check it right, but why do you check it? I can call this simulation time by t sim ok, but you have to define what is t sim.

(Refer Slide Time: 15:48)

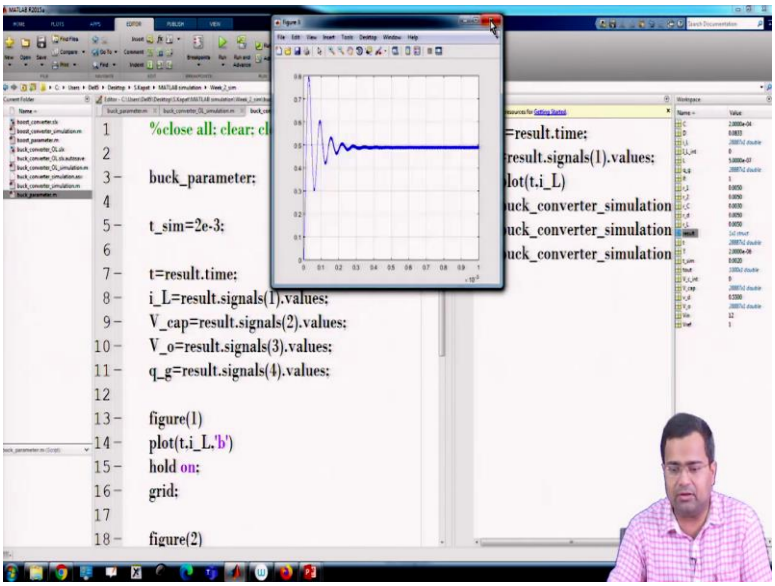


So, I can run this t sim; t sim for 2 millisecond duration ok, it will run.

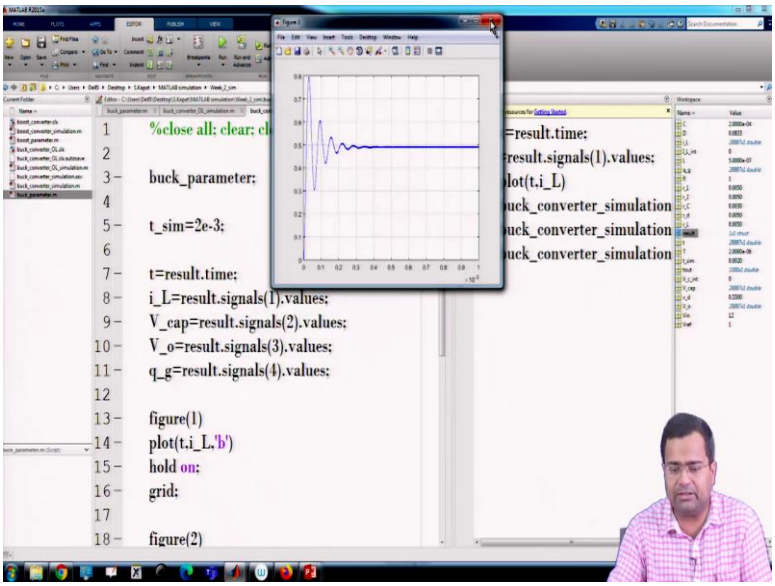
(Refer Slide Time: 16:00)



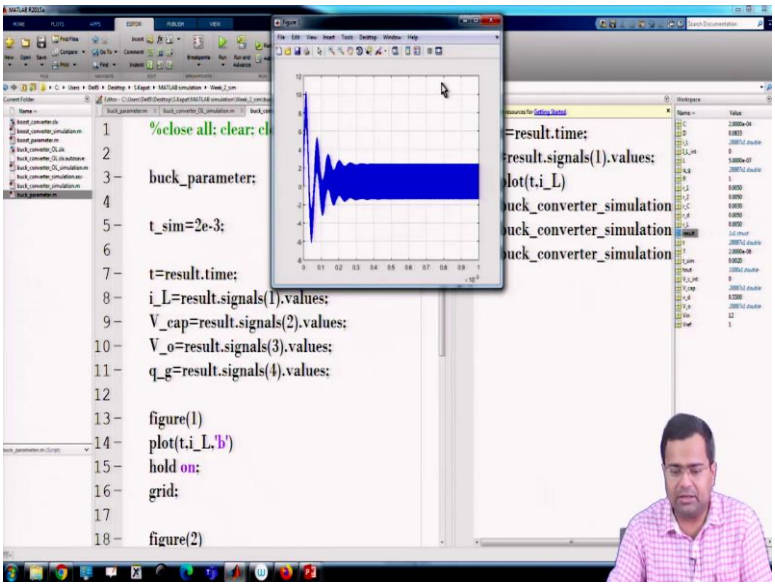
(Refer Slide Time: 16:02)



(Refer Slide Time: 16:02)

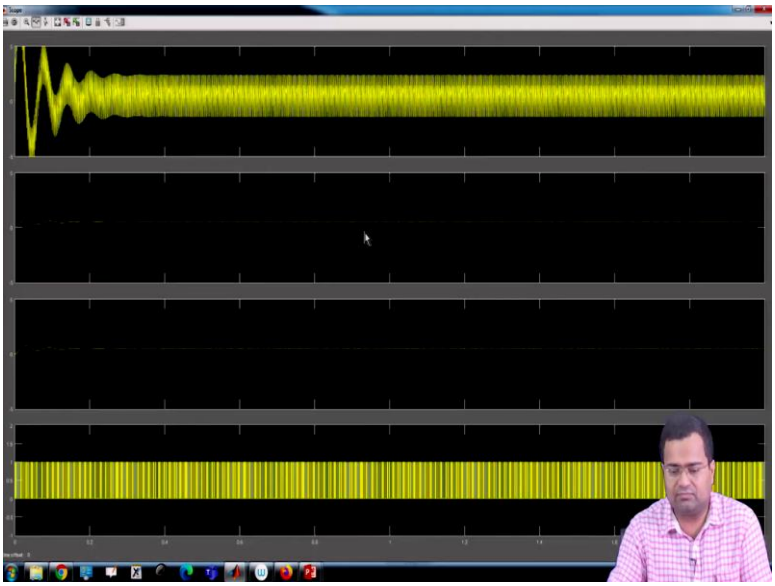


(Refer Slide Time: 16:03)



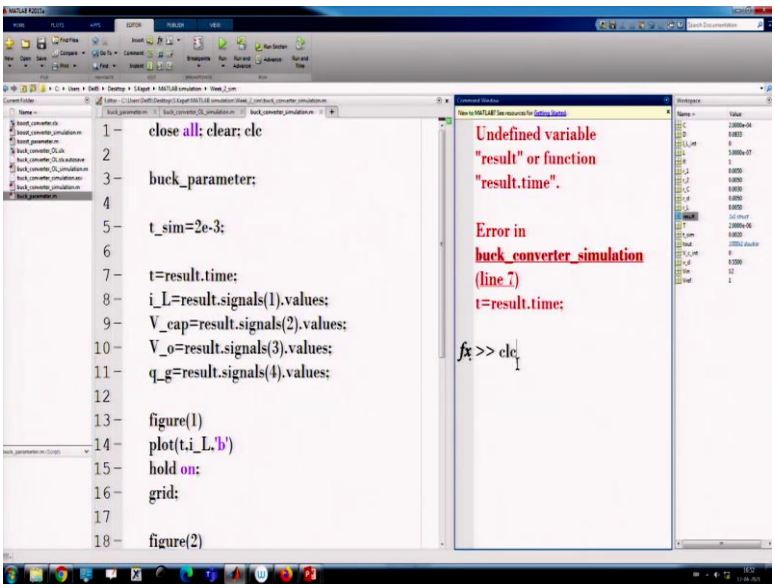
Once it will execute that means, but you have only set t sim so, you have to physically run it.

(Refer Slide Time: 16:09)



So, it will run for 2 millisecond now right. Then, how do you change the resistance value ok?

(Refer Slide Time: 17:17)

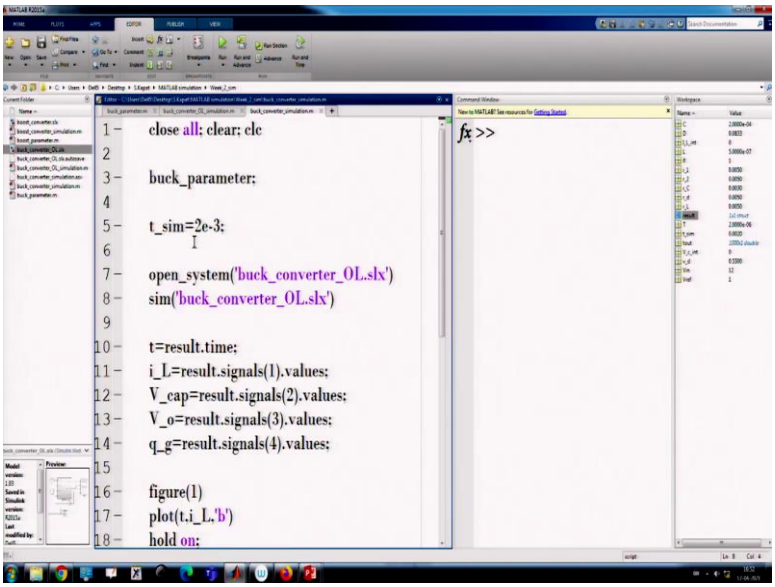


Next question now, having doing all these thing, you have to go back and forth to Simulink file to changes, check there, take the values, why should you do that? First of all, if you want to plot in the Simulink that itself will take time because Simulink plot while it is running and plotting together, it will actually takes more time, it will actually

take more time then if you run it and plot it after the simulation is over, you can store it so, this is possible.

Now, that means, what we what should I do next? Solve this figure plot. So, what we discussed first? We can customize this thing you know the simulation time, we can call the results separately and then, we can plot, but once we enable this block, then you cannot run because now, it close everything so, you have to again go back and run the simulation otherwise, the data's are not. So, you have to run the simulation and that is itself is a cumbersome job.

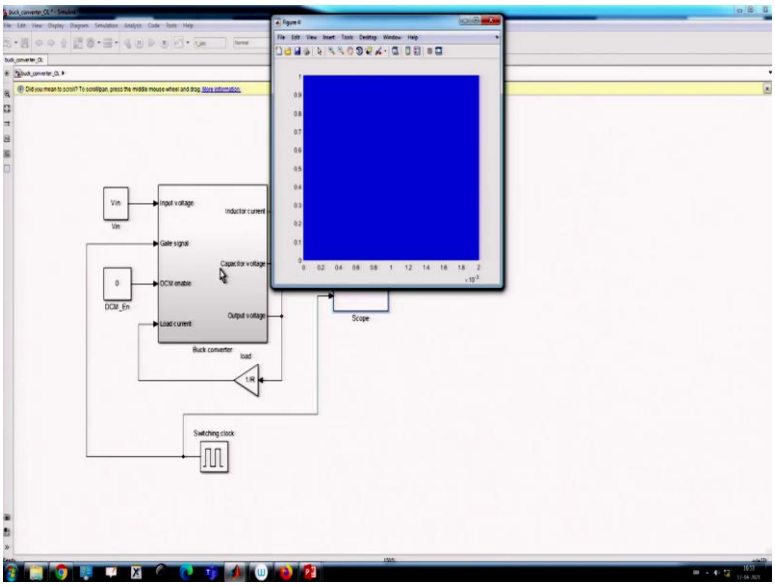
(Refer Slide Time: 17:44)



So, why should I do that? So, to save that, I should call the simulation right from my dot m file. That means, I can just run open system, then I can run the Simulink file like this. You have to provide the Simulink file name. What is the Simulink file name? So, this is the name of the Simulink file.

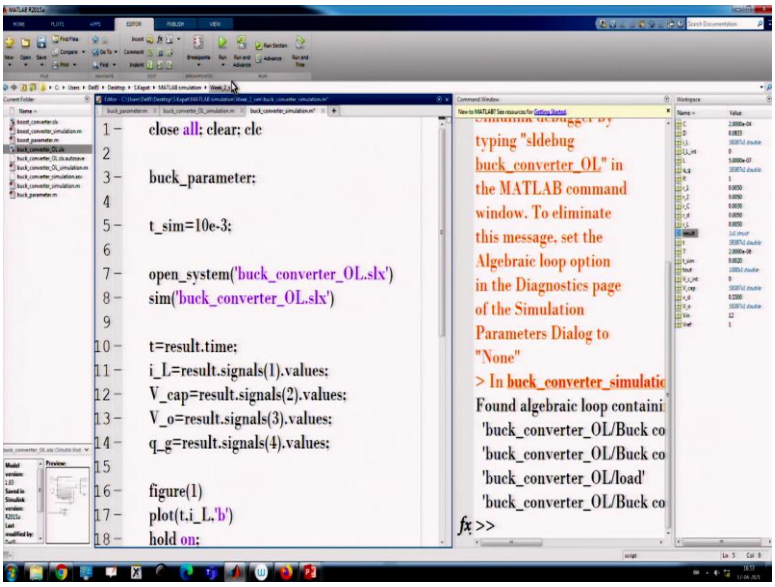
So, I am running this open loop buck converter this xls file. So, I have to provide dot slx; slx this is one. The next part I need to simulate, I want to simulate calling from here itself. Now, I do not have to go to my you know Simulink separately.

(Refer Slide Time: 18:30)



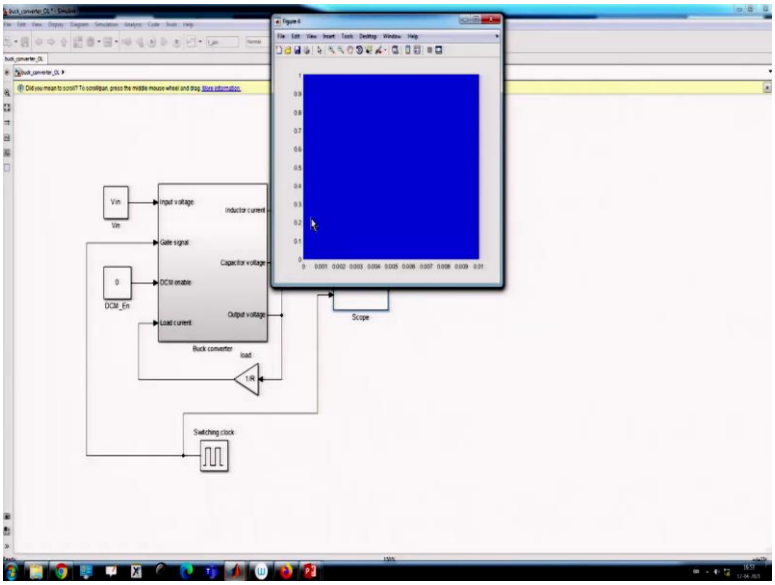
So, I can simply run it from here, it will run and again plot all these.

(Refer Slide Time: 18:39)



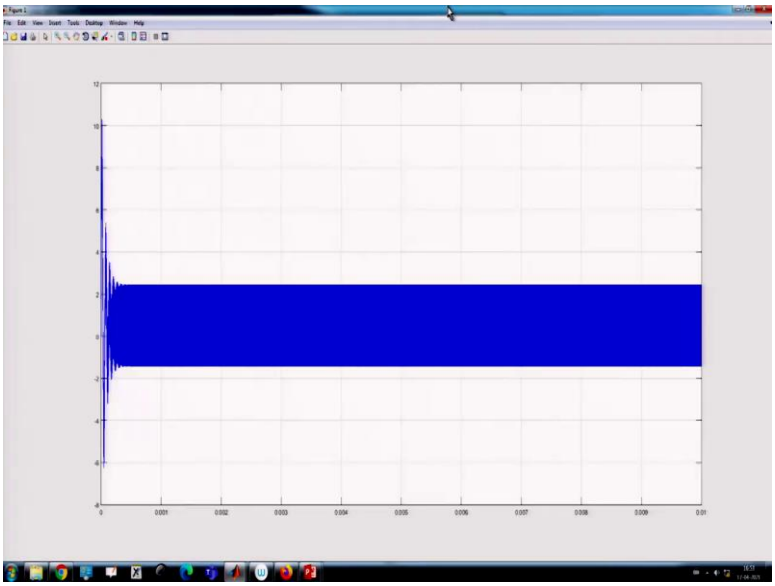
That means now, if I change my simulation time from 1 millisecond to 10 millisecond, my everything data, all these will be change automatically.

(Refer Slide Time: 18:47)



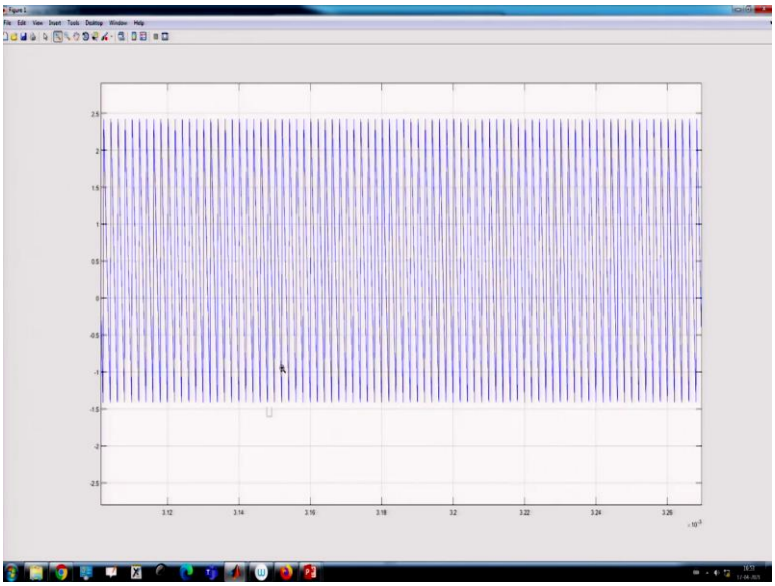
So, it is running the simulation for 10 millisecond.

(Refer Slide Time: 18:50)

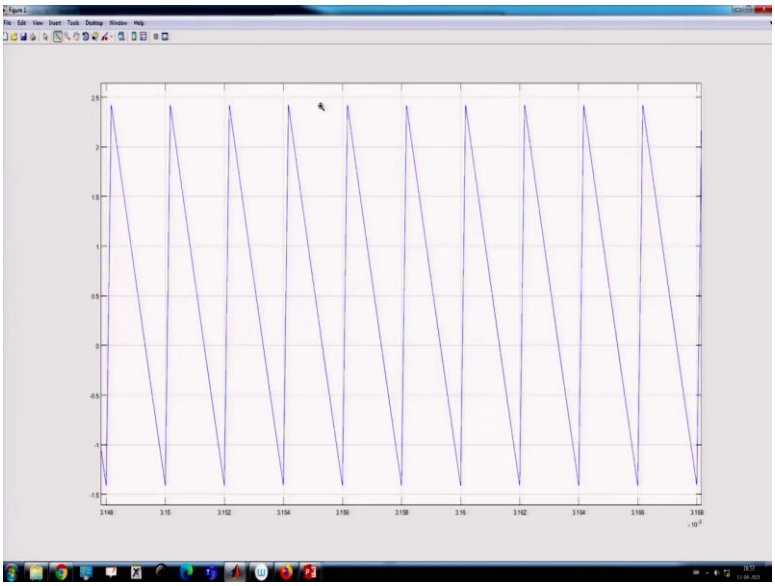


And plotting all these current waveform for 10 millisecond which is 0.01 ok alright.

(Refer Slide Time: 18:58)

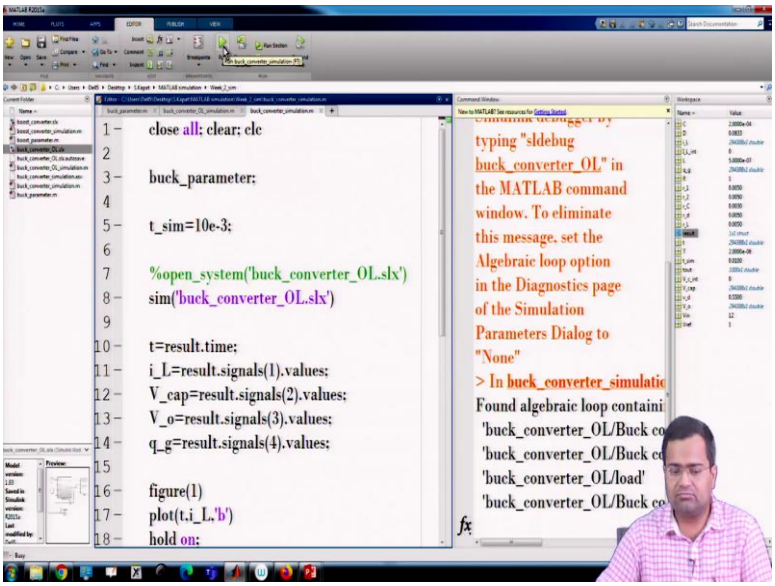


(Refer Slide Time: 18:59)



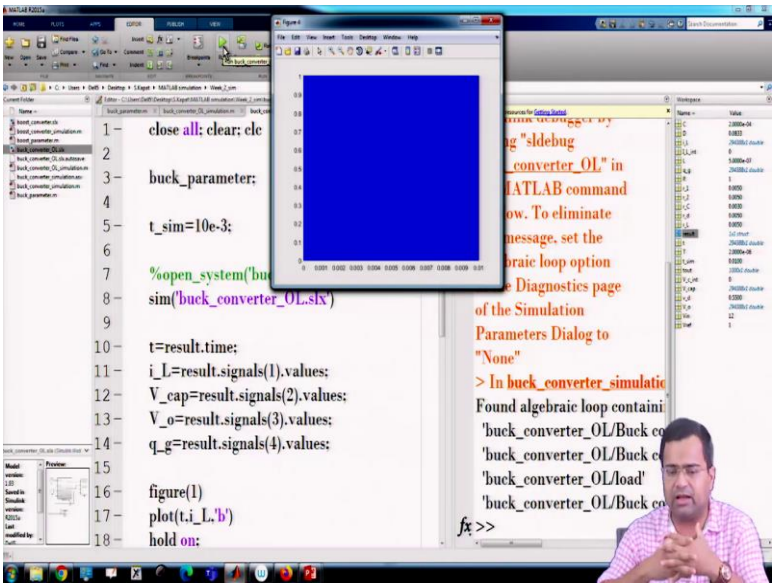
So, you can see the current waveform from here, you can check all these scope just by running simulation from here.

(Refer Slide Time: 19:21)



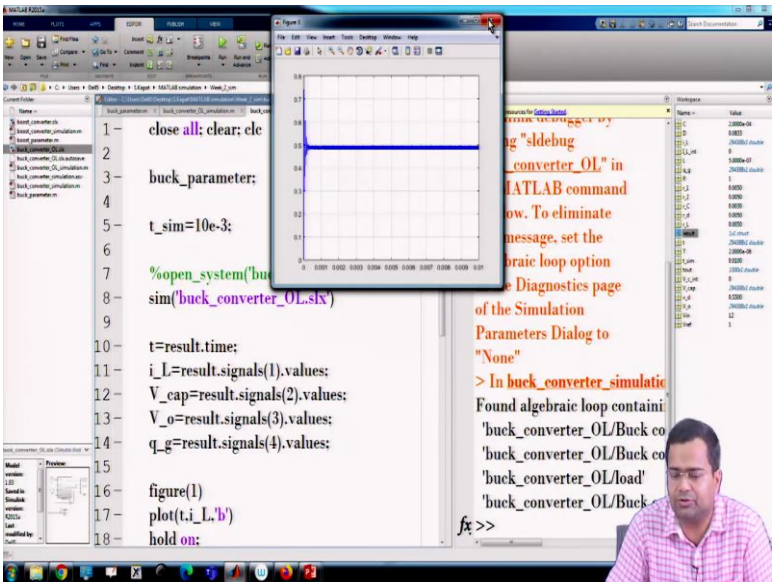
That means, what we learn? That means, we can call the simulation file, Simulink file from the script file ok. Even if you do not want to open, that is also fine. It will just run the simulation file and show the results.

(Refer Slide Time: 19:28)

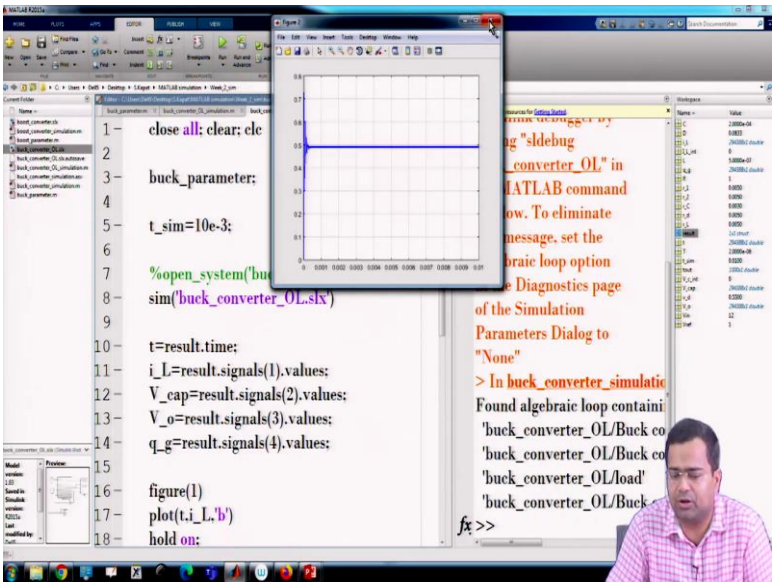


It will take some time to simulate and show the results.

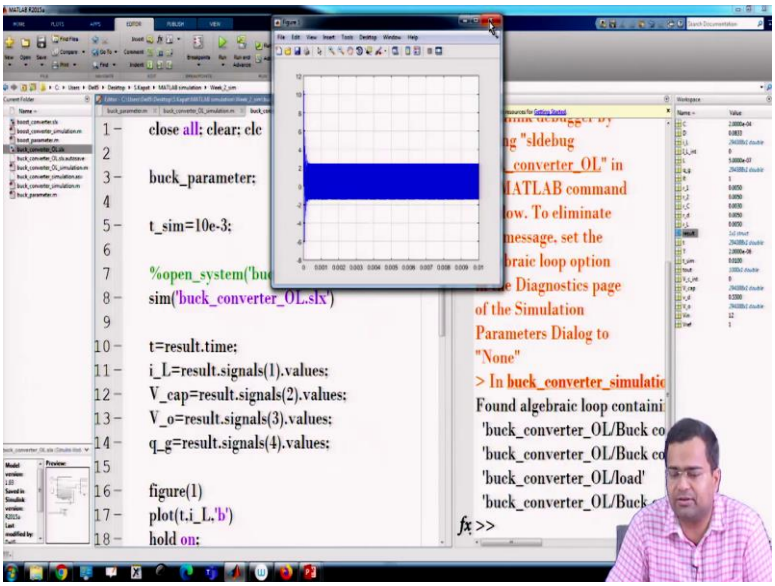
(Refer Slide Time: 19:31)



(Refer Slide Time: 19:31)

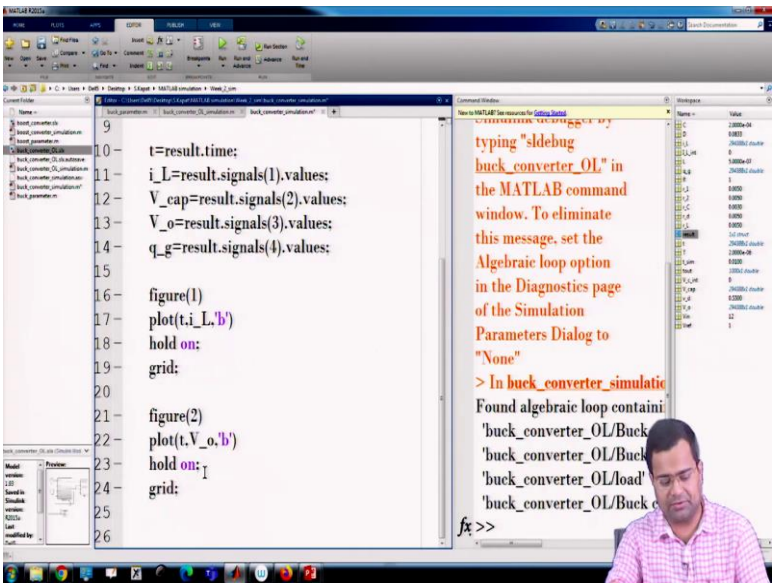


(Refer Slide Time: 19:32)



So, all four results will be shown one by one. So, you can enable or disable, and it does not matter. Our next task is to customize the simulation time. I have called the parameter file separately at the beginning; I have to call the Simulink file from here; I have run it. This is for executing the simulation so, that means, when you call this, it will run the Simulink file just by executing this command.

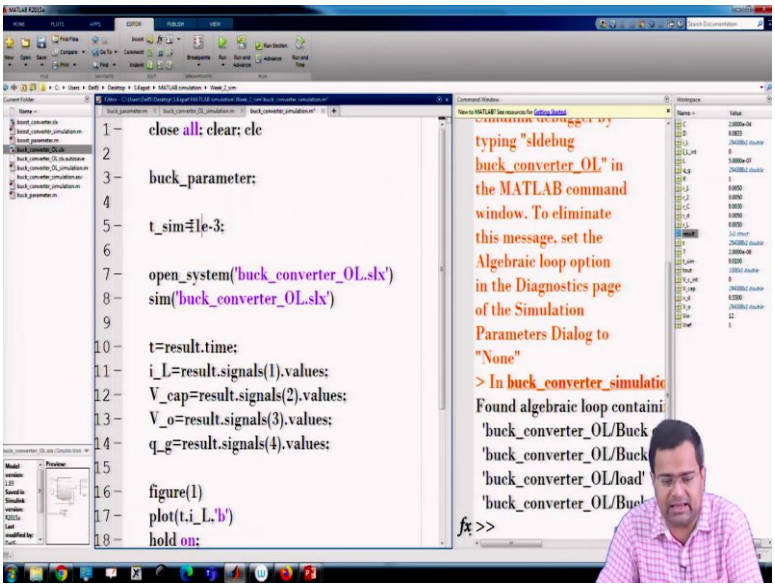
(Refer Slide Time: 20:22)



Then, I have stored all the results like a time, inductor current everything from here ok and you we have all these plot command, we can plot it. So, maybe we do not need to

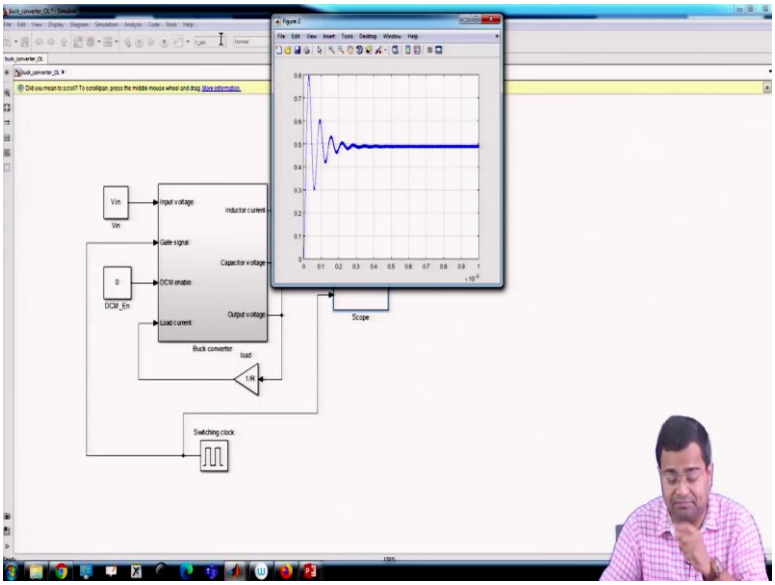
plot all these, first two it should be sufficient. We want to just plot output voltage and that is it, yeah. So, only two plots is sufficient. So, it will call t-sim is now running for 10 millisecond.

(Refer Slide Time: 20:40)



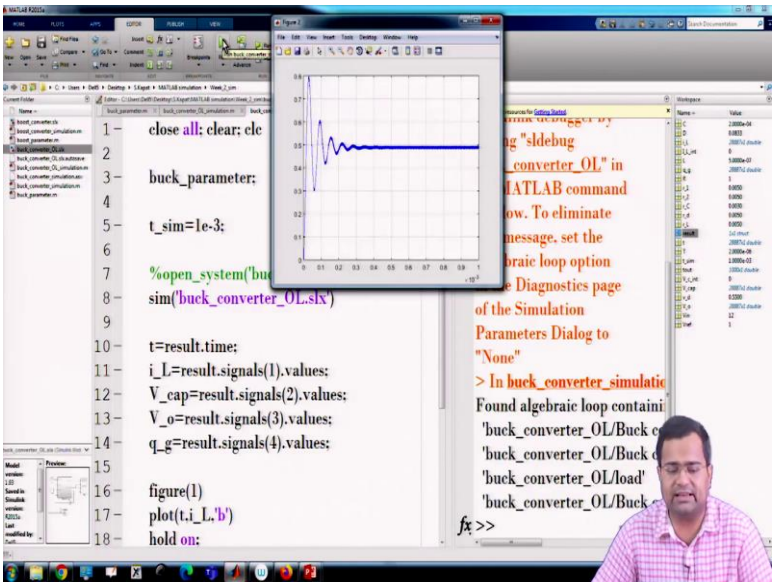
So, if we want to speed up simulation, then we need to, we can decrease to 1 millisecond and run it again.

(Refer Slide Time: 20:43)



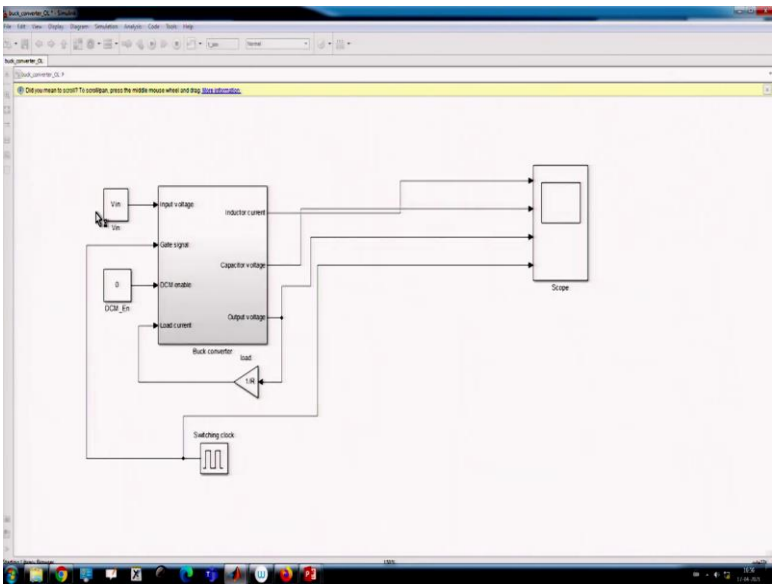
So, we can do whatever we want nice.

(Refer Slide Time: 20:55)



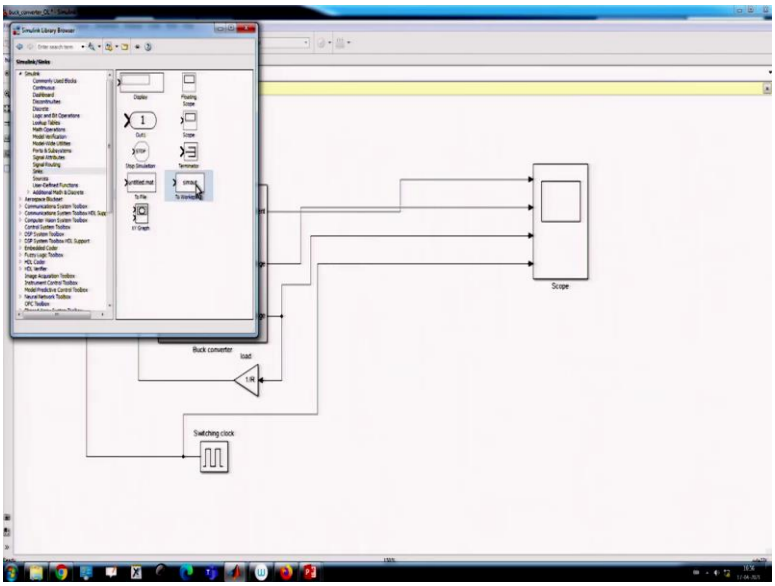
Now, why again we want to see all the time the Simulink file, if I just close it, it will not show go to that screen, it will run it.

(Refer Slide Time: 21:22)



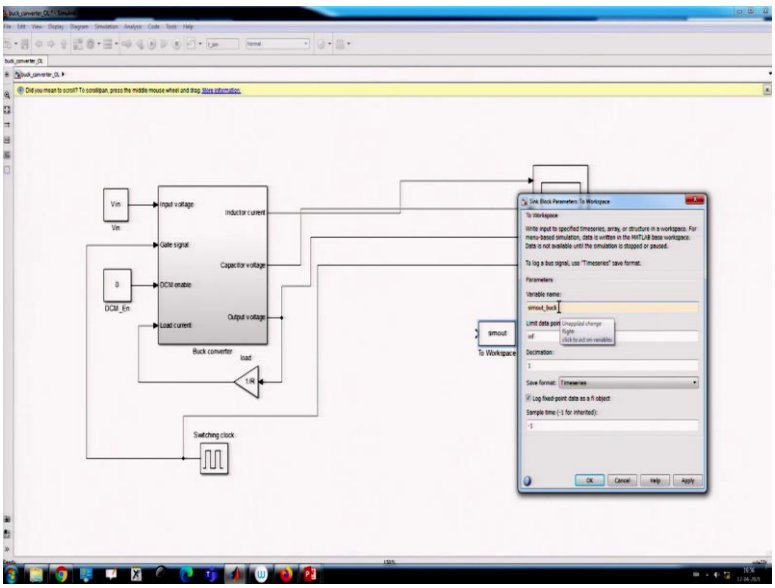
Now, I have a next task why should I use because we are calling the data into our workspace, then why do we need to again place a scope there? Because as I said if we want to display the scope during live like a when it is running, it will actually slow down the simulation so, we can in fact, disconnect the scope, we do not need this scope at all.

(Refer Slide Time: 21:34)



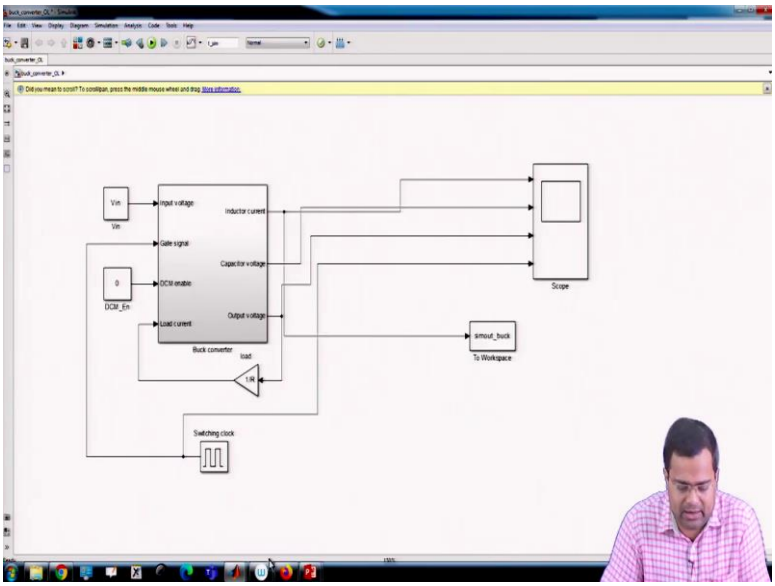
Or maybe initially before you that, I can use something like if you go to the sink block in the library, there you will find something like a simout block.

(Refer Slide Time: 21:39)



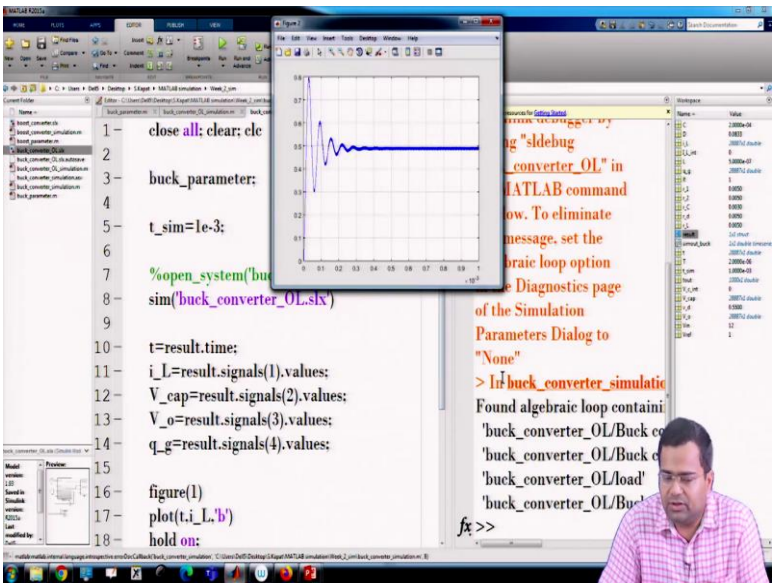
I want to take the data to the workspace. So, I do not need to pass through the scope so that I do not want to display. But this simout name can be change. You can use simout 1, 2, 3 whatever, it is your customize so, I am just using default ok simout_buck I call it ok so, simout.

(Refer Slide Time: 22:01)

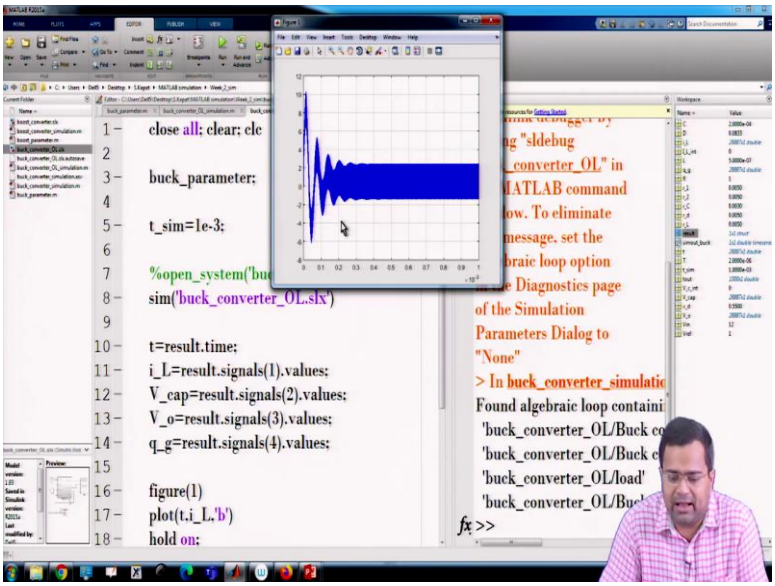


Now, I can use simply buck no problem anything that we want, but it has only one input whereas, I need to take various signal data to the workspace. So, that means do you need multiple such simout block or we can use a vector input for the simout that is possible. So, if we have only one inductor current so, this is my inductor current. Now, what I am doing? I am taking the simout and now; I am just running it.

(Refer Slide Time: 22:43)

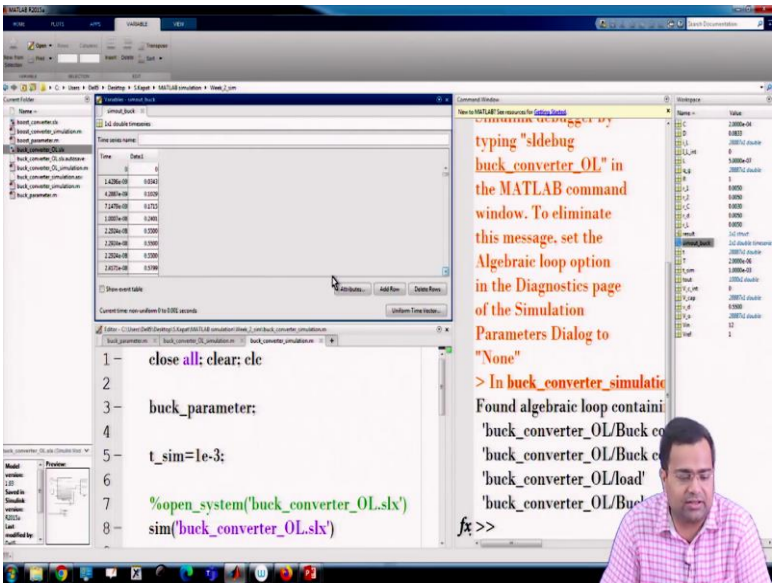


(Refer Slide Time: 22:46)



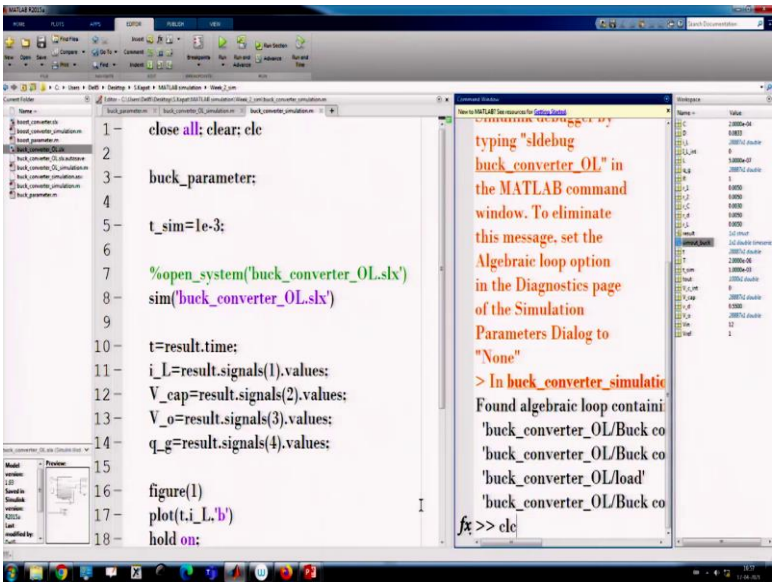
If you run it see in figure 1, you have plotted inductor current. This current is coming from the result block which is connected to this particular scope. I want to check that sim out data of the inductor current should be identical.

(Refer Slide Time: 23:04)

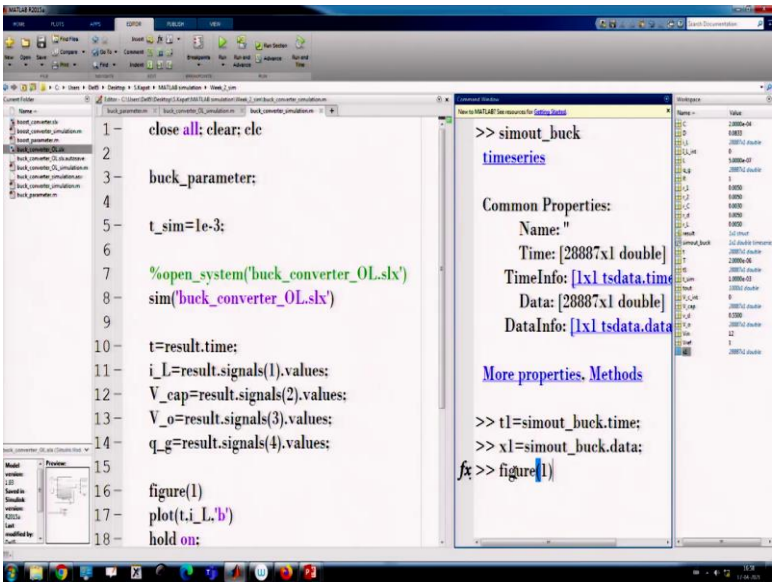


So, you go to simout, this is also structure. The first is one time and the second is data ok. So, first is time and the second is data.

(Refer Slide Time: 23:19)

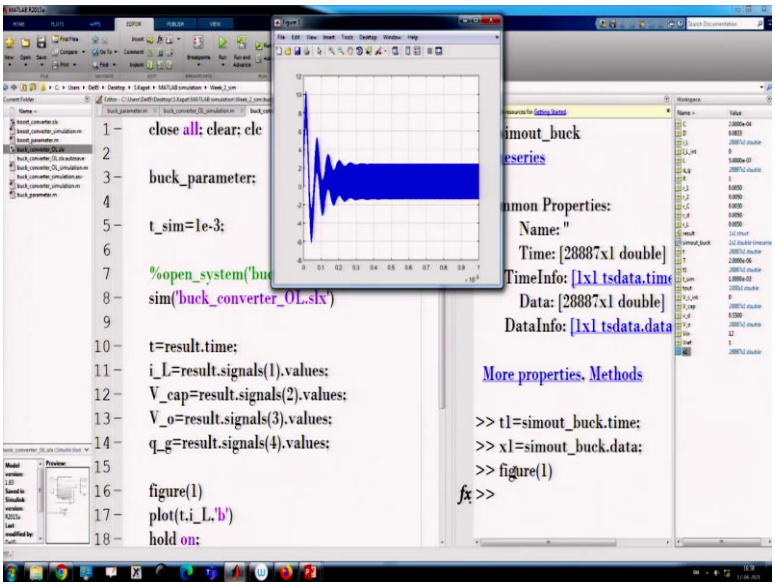


(Refer Slide Time: 23:21)

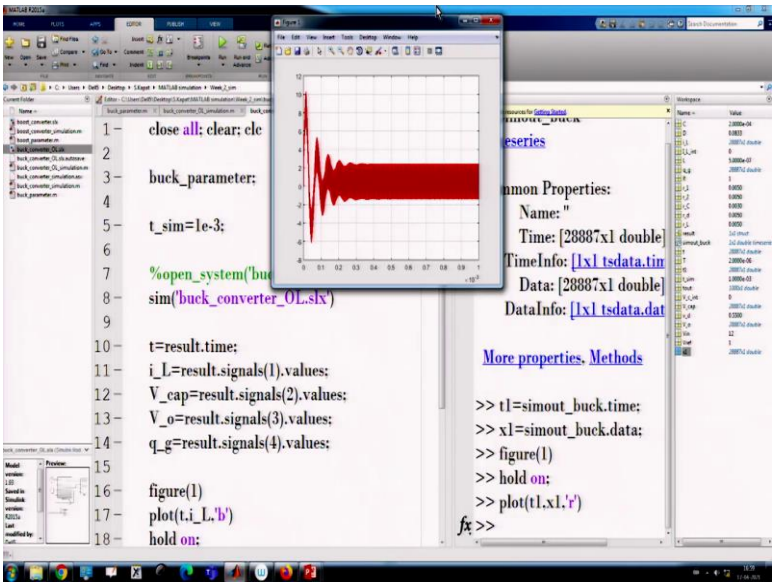


That means even if you write, what is my simout say that means, I want to just type this particular name ok, you just type it here, you will file the detailed description. So, it has a time vector and then; it has a data element ok. So, the time that means, if I write again t1 which is equal to simout dot time, t1 is now created and if I write let us say x1 which is simout dot data. Since x1, since we are using only one like a scalar input so, it will directly give vector. Now, I want to plot, let us say I want to call the figure 1 ok.

(Refer Slide Time: 24:11)

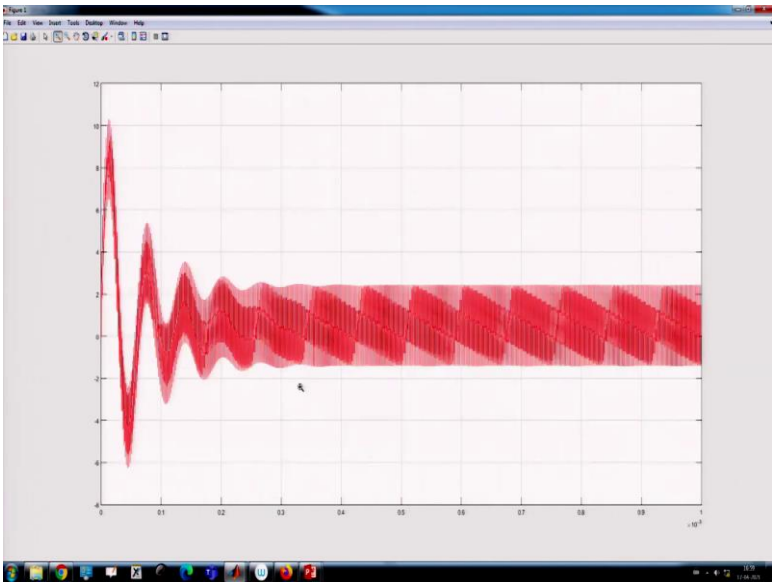


(Refer Slide Time: 24:13)



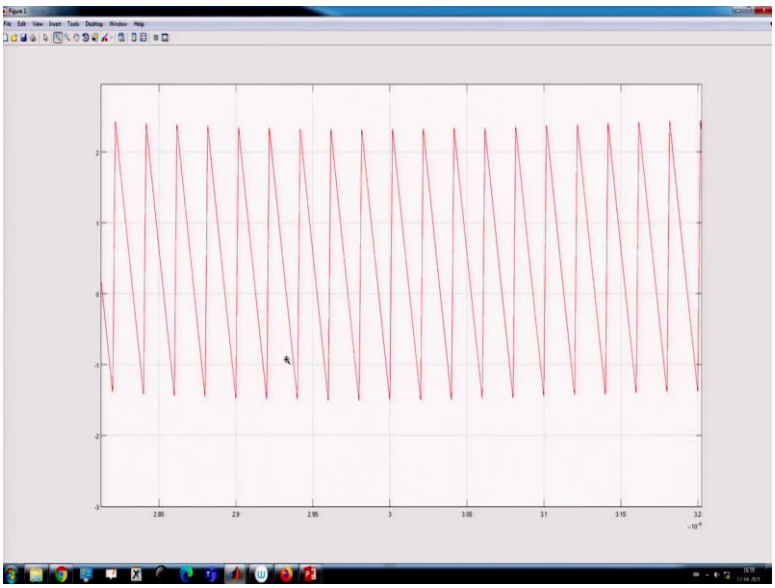
Then, hold on, then I want to plot, plot t1 comma x1 and I want to use a red colour on that.

(Refer Slide Time: 24:32)

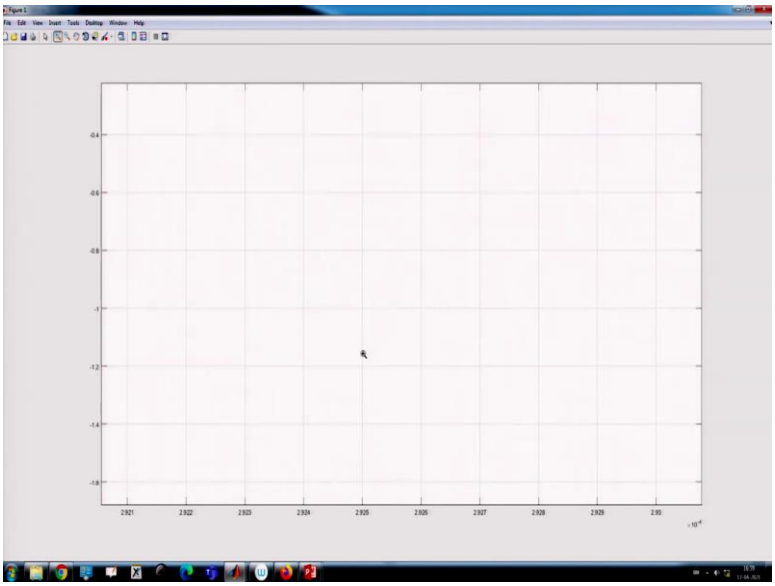


Now, if you see both are exactly same so, you cannot distinguish. Both are like a blue and both are merged together.

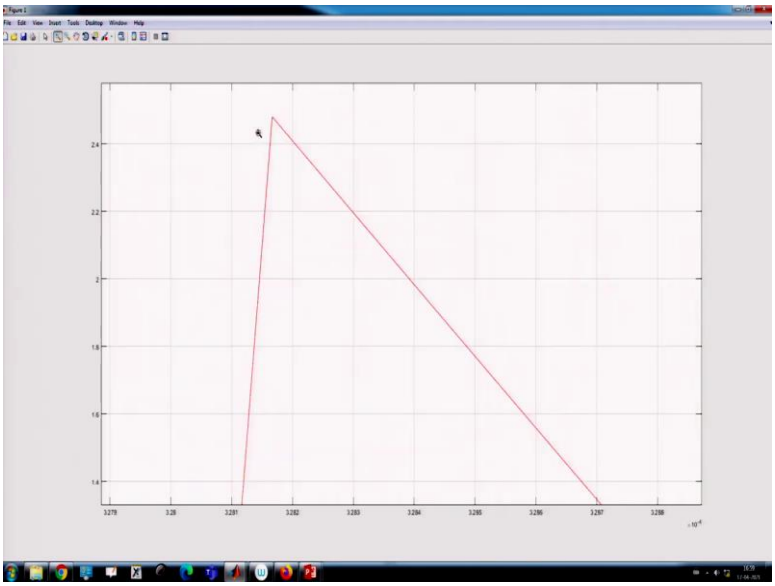
(Refer Slide Time: 24:35)



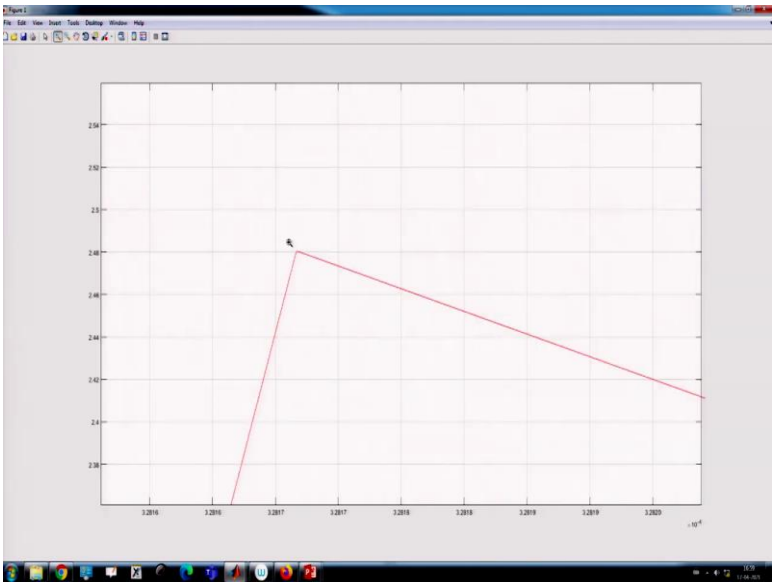
(Refer Slide Time: 24:36)



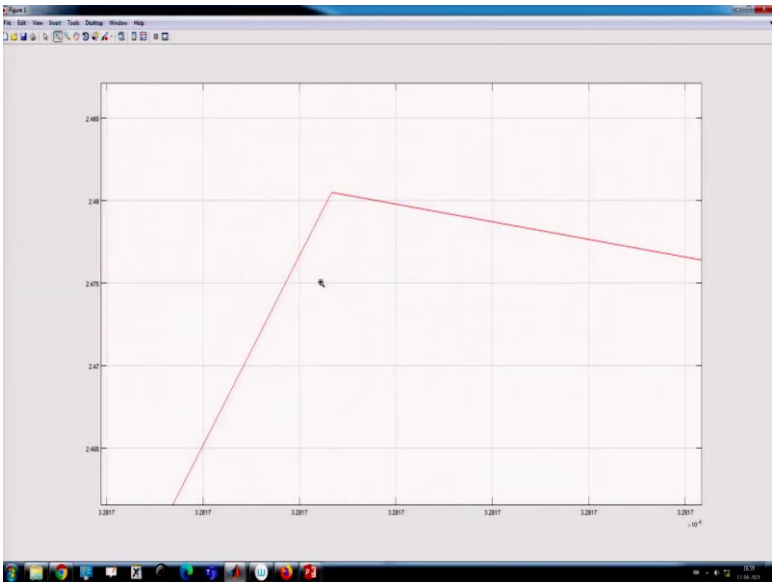
(Refer Slide Time: 24:41)



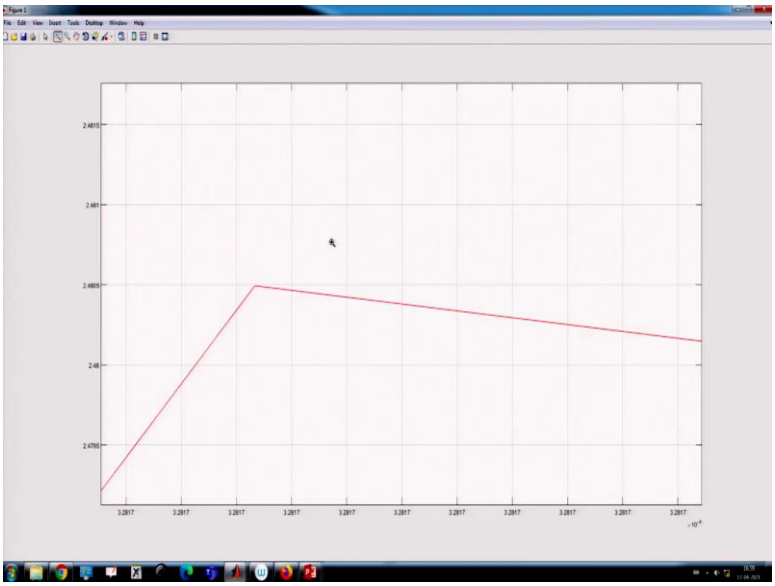
(Refer Slide Time: 24:42)



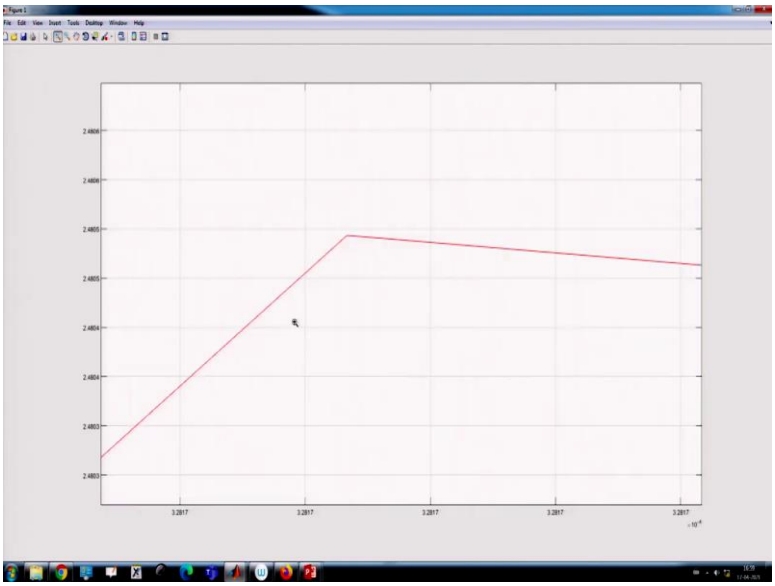
(Refer Slide Time: 24:43)



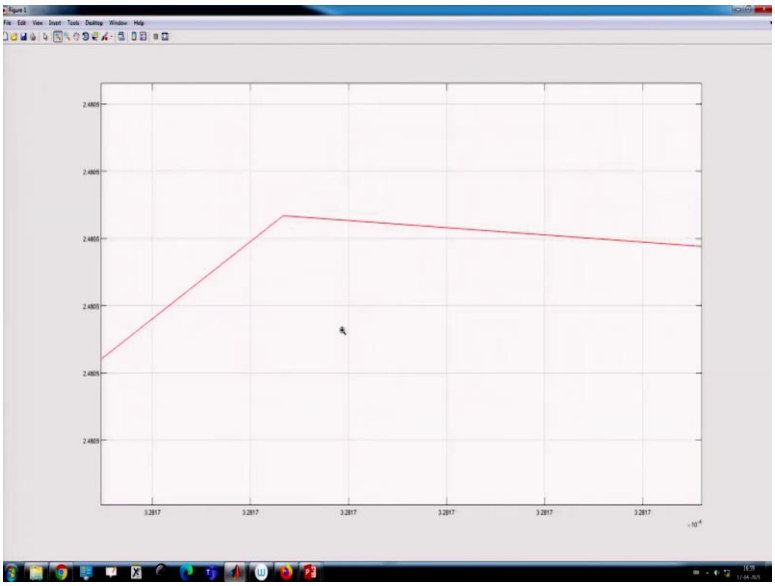
(Refer Slide Time: 24:44)



(Refer Slide Time: 24:45)

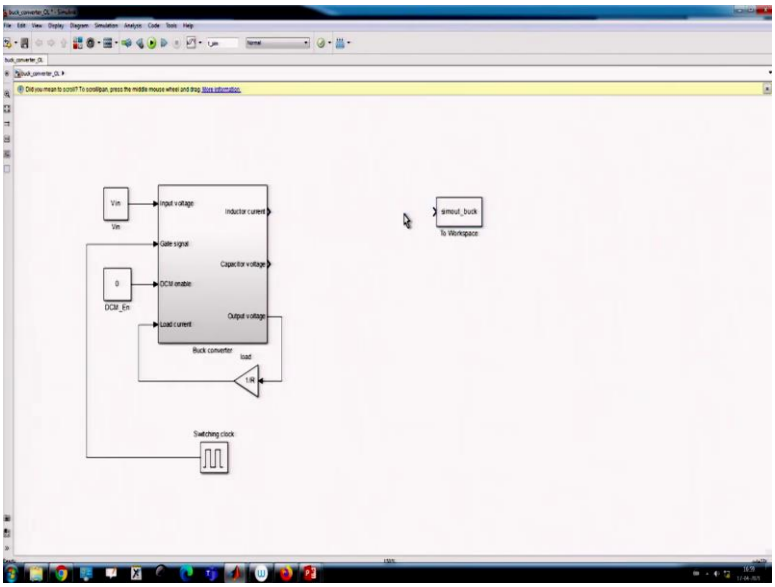


(Refer Slide Time: 24:46)

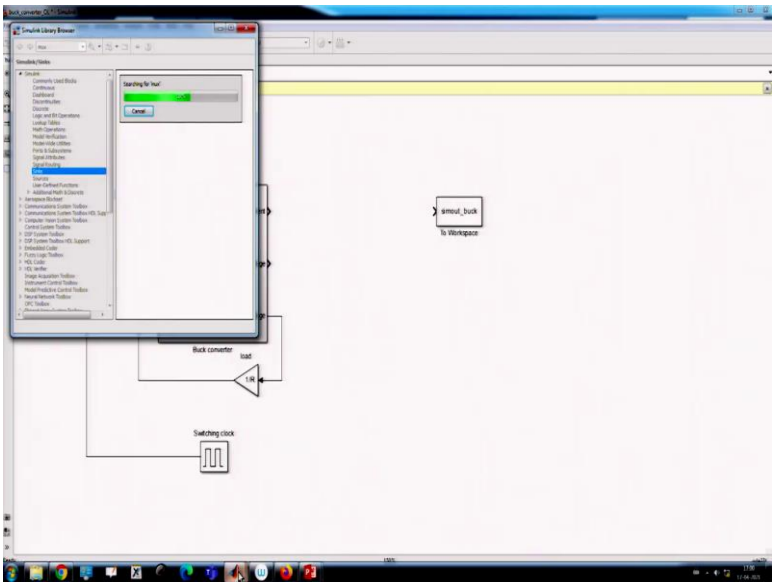


So, you cannot identify that because they are the same data, same exactly same data. So, it is not both are like overlap data right so, you cannot distinguish them. That means, I do not need to use a scope to bypass the data. So, I can remove this scope completely.

(Refer Slide Time: 24:57)

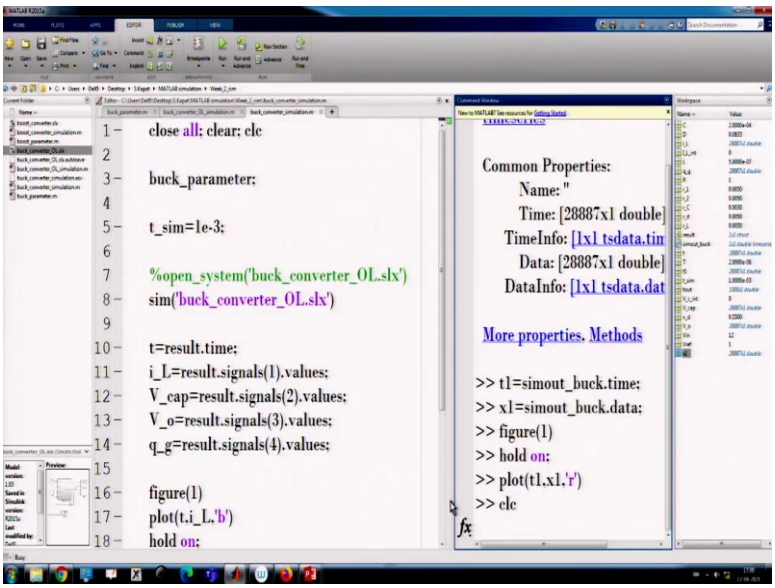


(Refer Slide Time: 25:03)



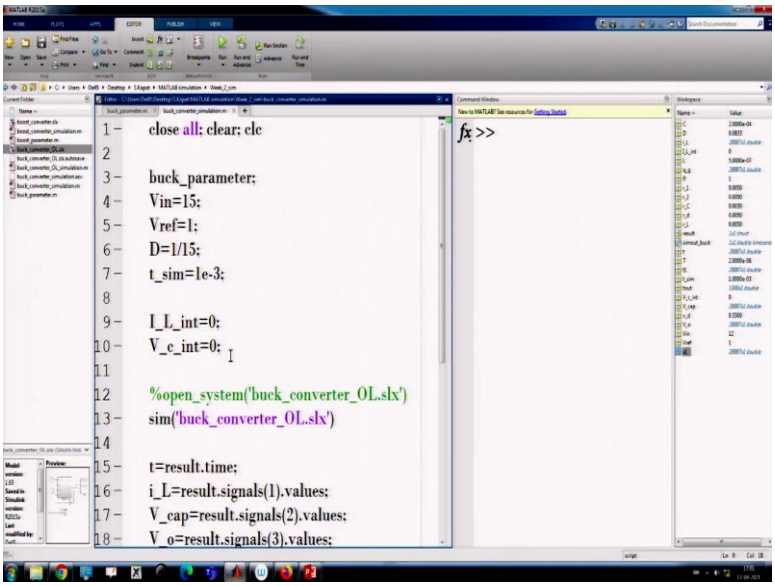
My next task is that simout, I should not use the scalar. I want to use a vector so, use a mux. So, let us use a mux. Now, once you have a multiplexer that means it is muxing multiple signal together so that you can take that data into workspace, multiple data at a time you can take it to the workspace ok.

(Refer Slide Time: 25:31)



So, before it I mean loads, so let us go and check ok.

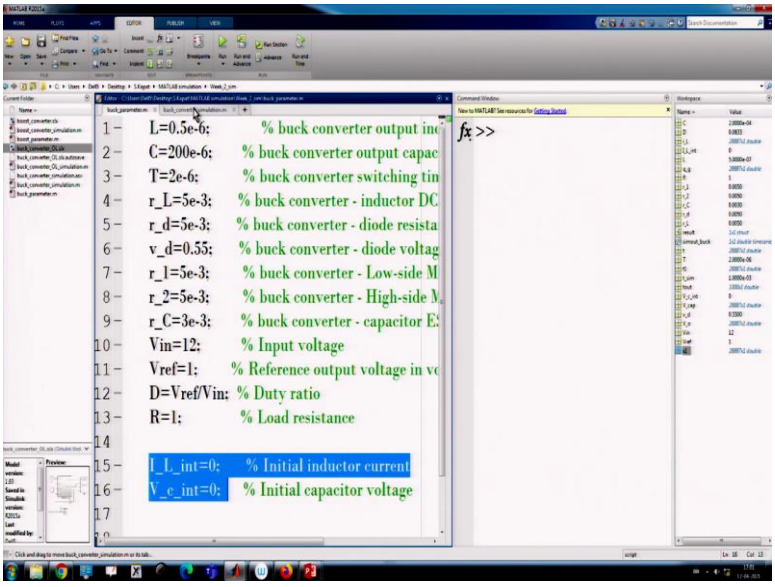
(Refer Slide Time: 25:32)



So, what we have seen that parameter file, after parameter, we will define something in fact, you can specify a different input voltage no problem, you can set it and you can set a different duty ratio like a 1 by 15 ok so that the different duty ratio in the Simulink file will be specified ok.

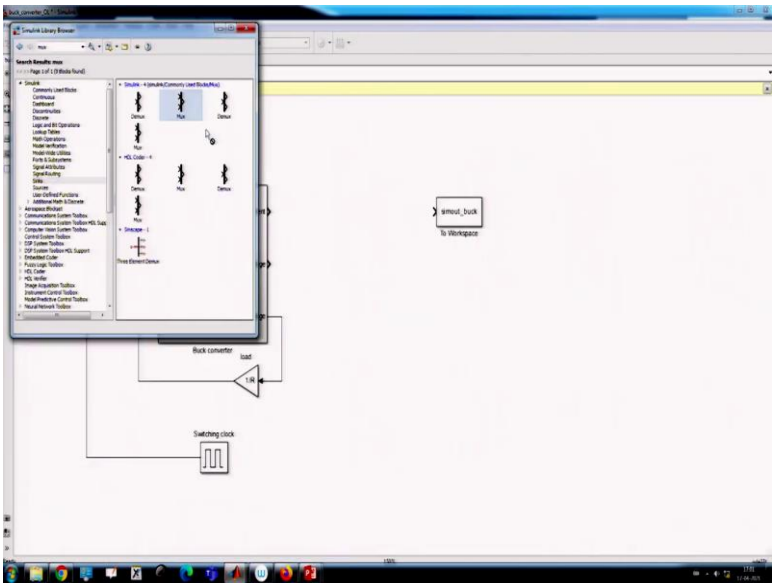
Next, if we go to you know Simulink, later it is waiting for the multiplexer to come. So, the duty ratio we can set, we can set Vref whatever we want a whether it is a parameter file, does not matter.

(Refer Slide Time: 26:32)



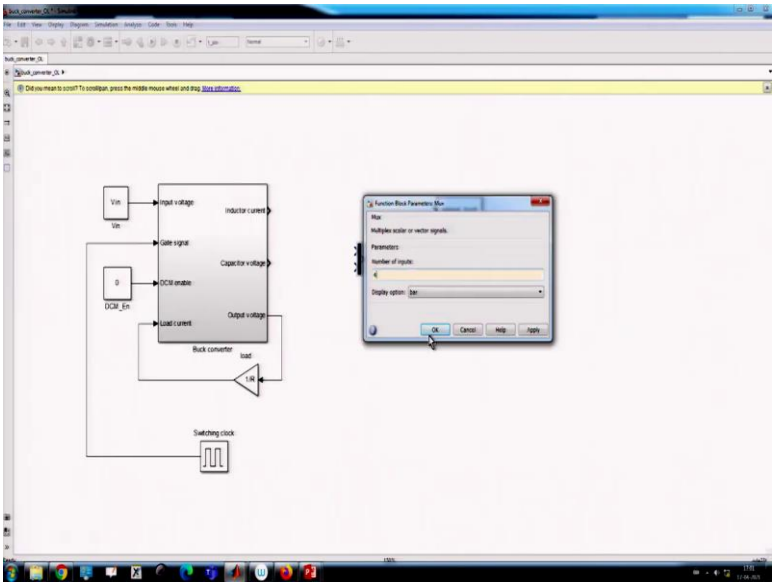
From the parameter file also, we want to call this initialization file as well. This is one of the most important part that we should specify at the beginning, initial value. So, these are the initial value of the inductor current and the output voltage ok.

(Refer Slide Time: 26:55)



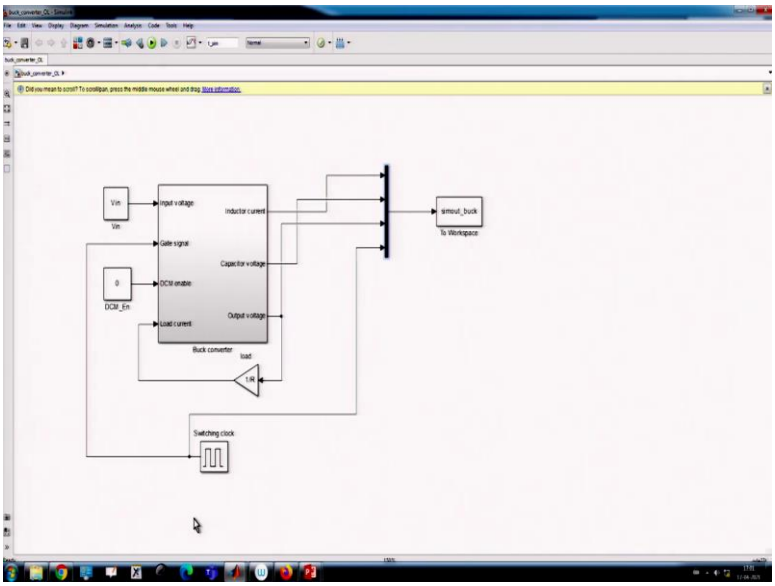
So, I can initialize from a separate point ok, let us go to the mux.

(Refer Slide Time: 26:58)



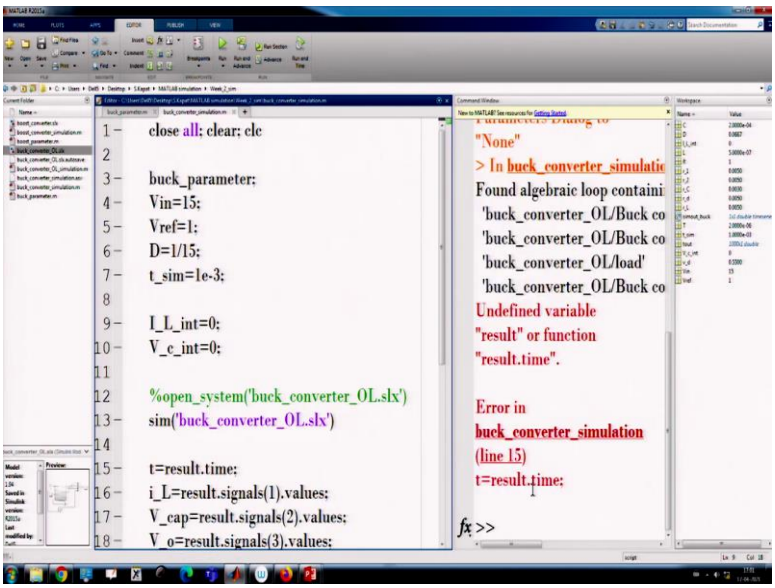
So, I can take multiplexer, I can take 4 output multiplexer ok.

(Refer Slide Time: 27:01)



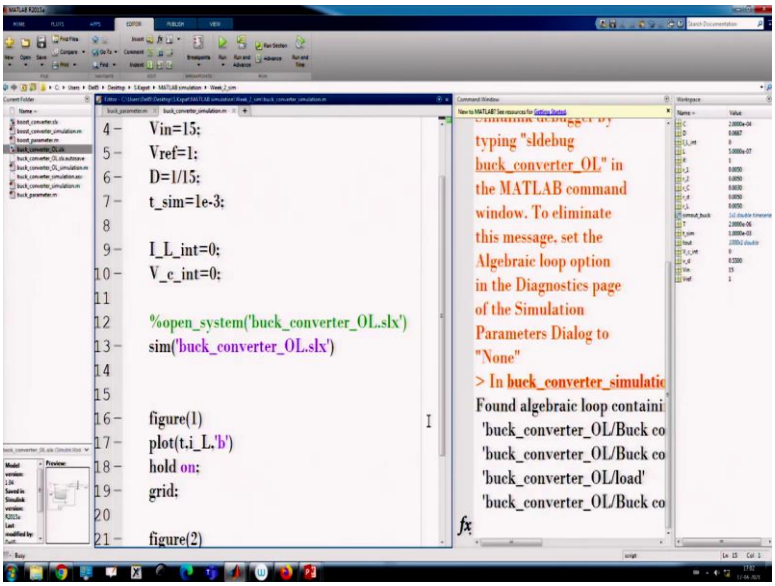
And it output is a vector. I can now connect to inductor current, capacitor voltage, output voltage and my gate signal all these are possible now. Now, let us go back.

(Refer Slide Time: 27:23)



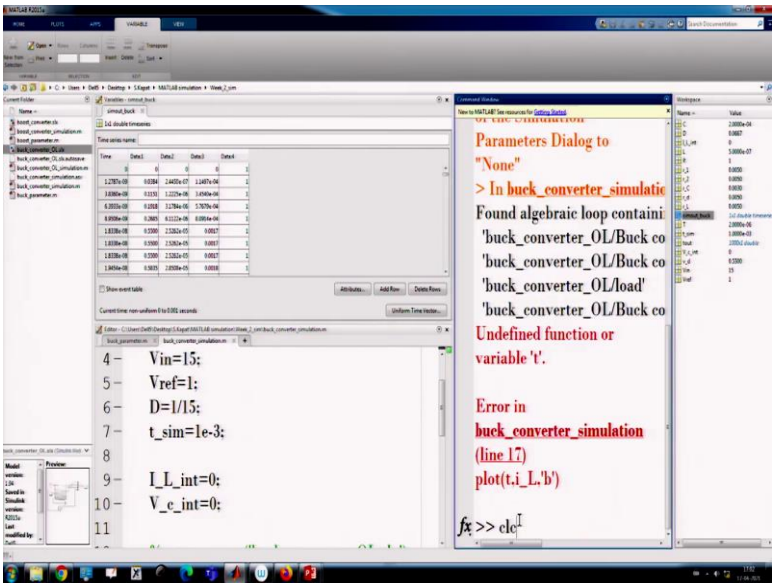
So, if you run now the result is not there so, we cannot run this.

(Refer Slide Time: 27:31)



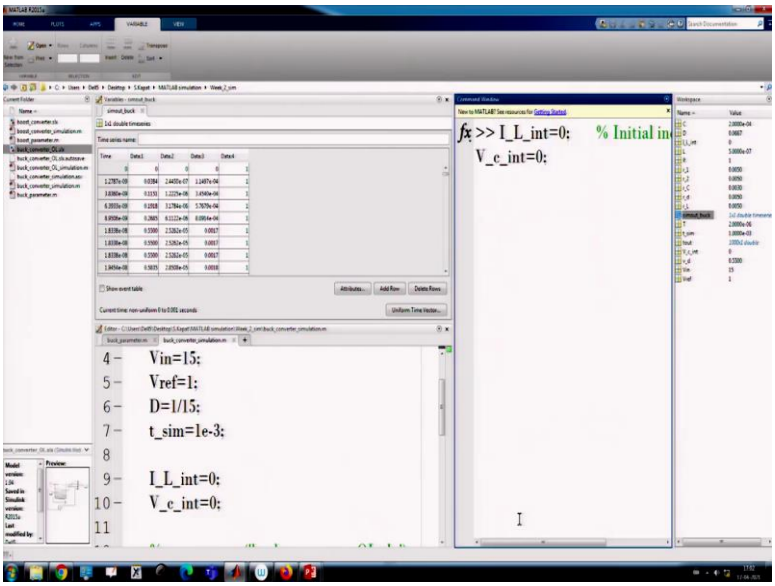
Because we have eliminated the scope ok, we have eliminated the scope, but we need to plot it. Where is time vector and all these we need to plot it. So, how to get this time? It will show error because you know it does not have the time.

(Refer Slide Time: 27:53)



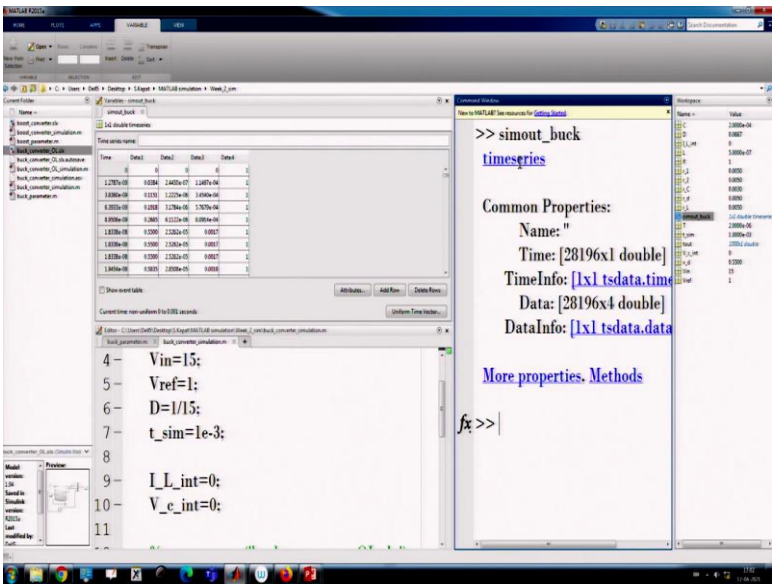
Now, what is there? The same buck we have stored.

(Refer Slide Time: 28:01)



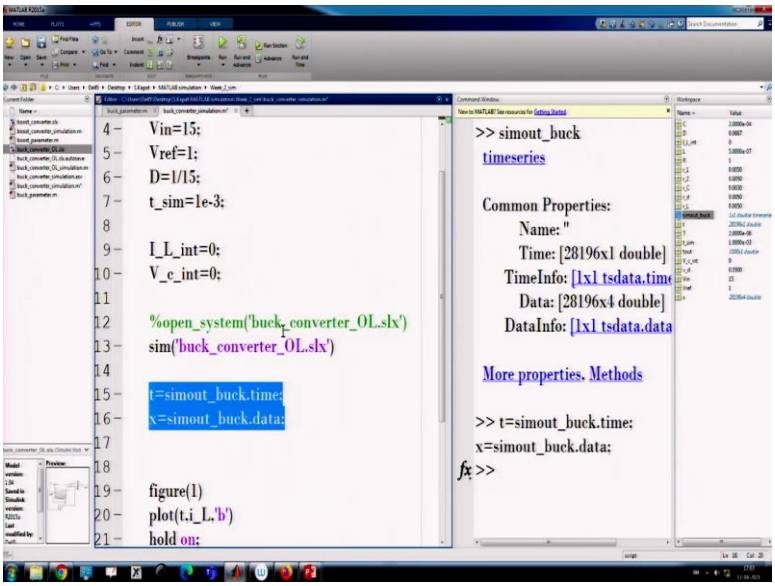
Now, if you go inside, it has 1 time vector that means, if I click here that means if I write just a minute sim buck.

(Refer Slide Time: 28:08)



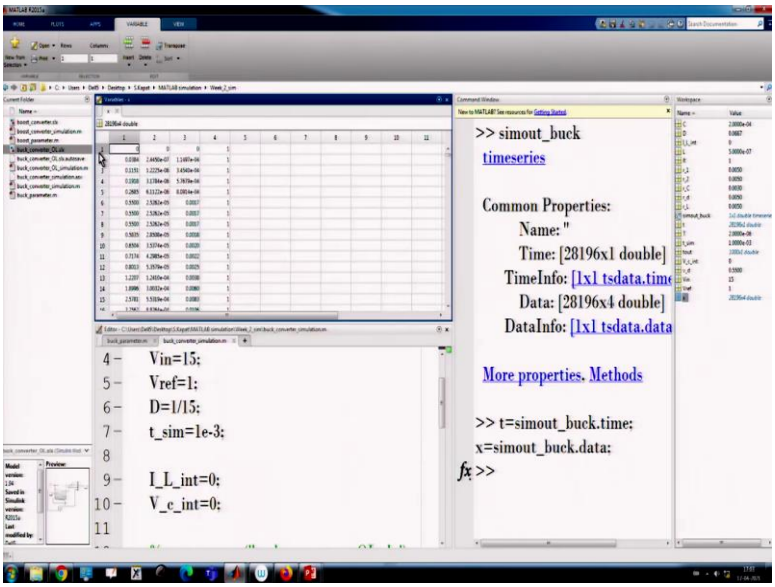
So, sim buck has one time vector, and the data is a structure that means. Each data has four columns, and each column has many elements that correspond to the values of that particular signal at different time instant ok.

(Refer Slide Time: 28:36)



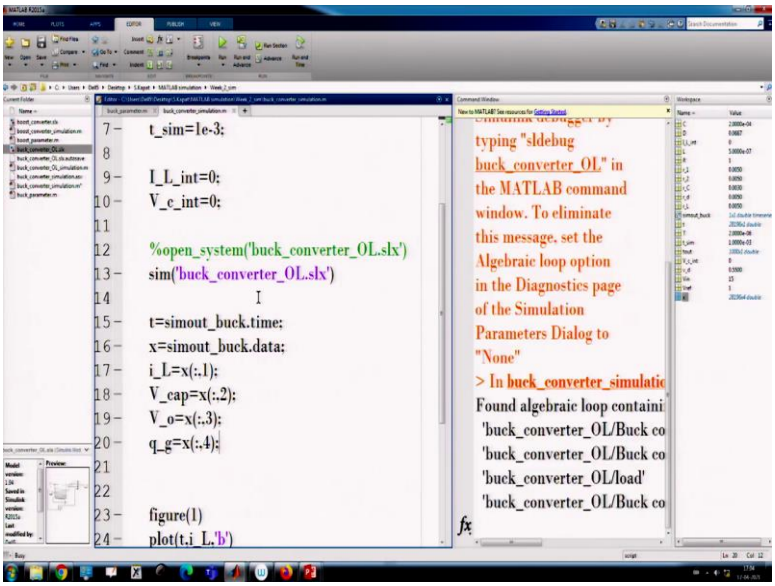
So, that means, now we are trying to do, we are first trying to create t equal to simout dot time and we are writing x simout dot data. So, this data we will take if you run these two, what I will first do? If you run these two.

(Refer Slide Time: 28:57)



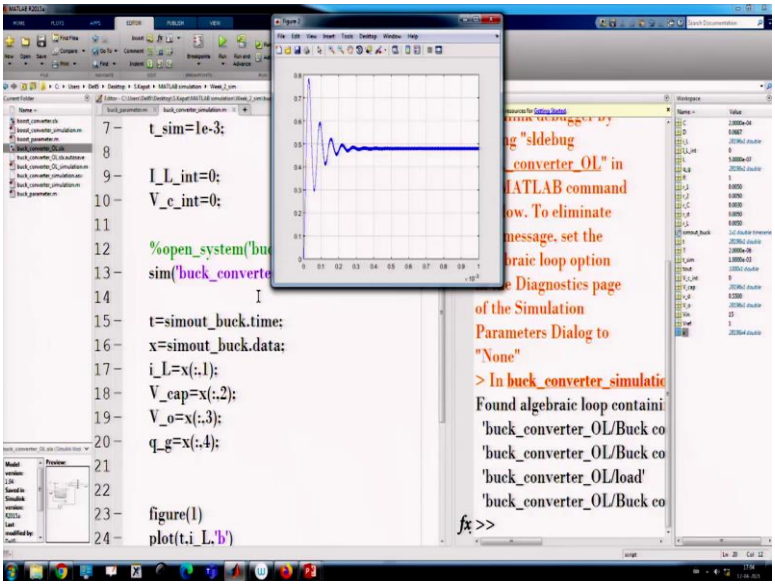
Now, if you go to x, x has 4 column that means, 4 column and each column you have data ok.

(Refer Slide Time: 29:05)



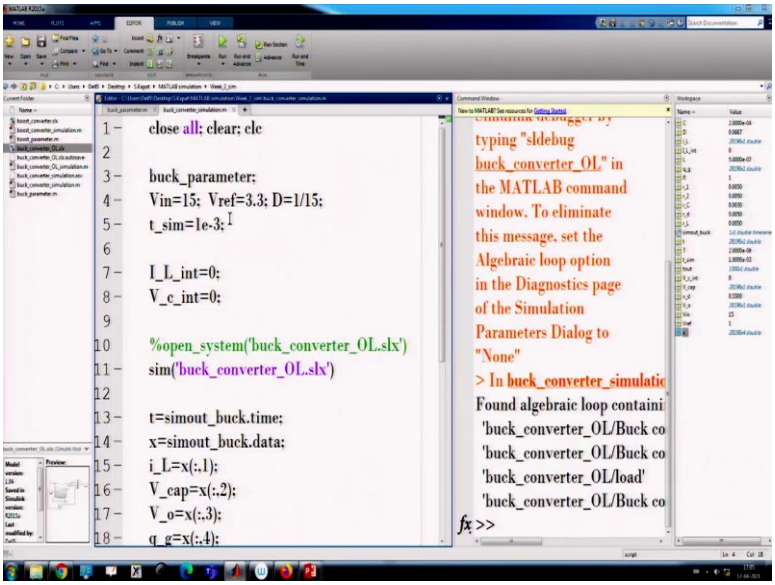
So, we have extracted. Now, our inductor current is actually in the first column, it is the first column so, x. So, all the data in the first column will take a row, all the rows will take and the first column will take. Next, what are their inductor current, then capacitor voltage cap, it will be the 2nd column; 2nd column, then V 0, it is the 3rd column and then, q g which is my 4th column. Now, once we do that now, we are ready and now, we can draw the plot ok.

(Refer Slide Time: 29:50)



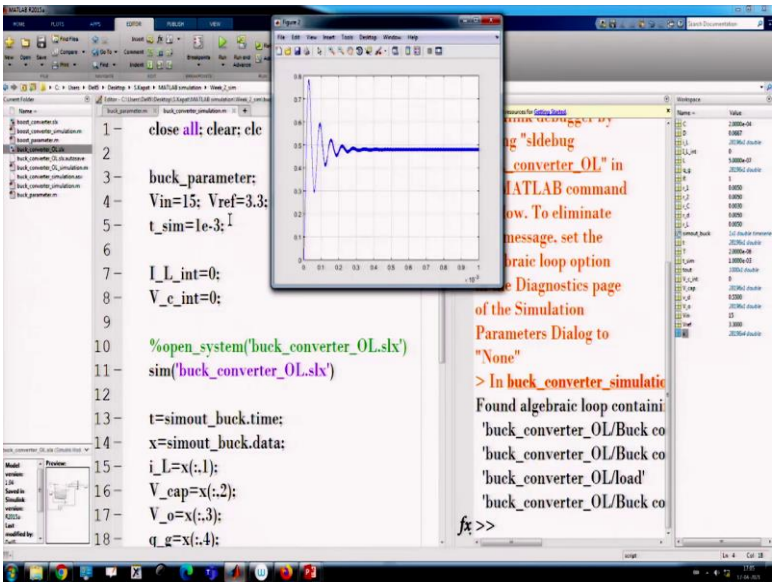
So, you can see now, all the data's are captured, the current and everything we got back just by there is no scope anymore so, that means, there is no scope and you can speed up the simulation ok. So, you have created a transient case here. We may or may not open this block does not matter. We can initialize here, and we can define different parameters here.

(Refer Slide Time: 30:19)

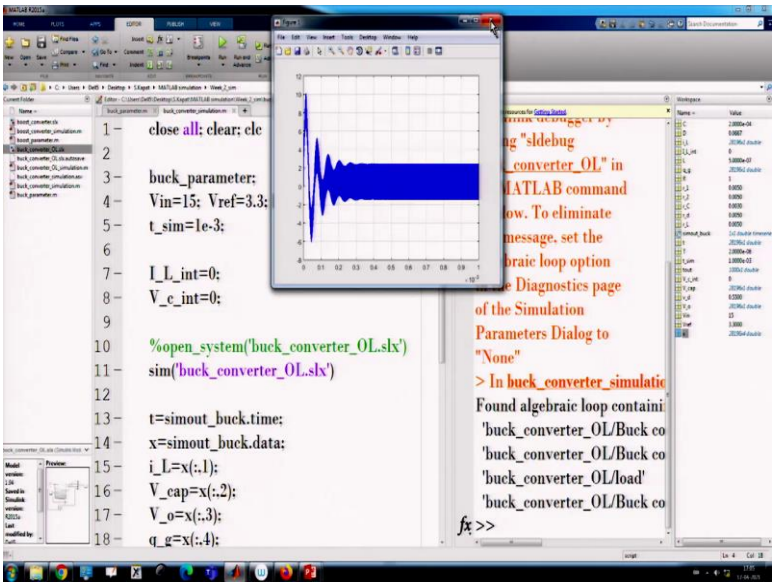


So, let us say voltage something here, buck parameter, we can define duty ratio. Now, if we change the duty ratio, let us say if we change the output to 3.3, we will see what happens, you run it.

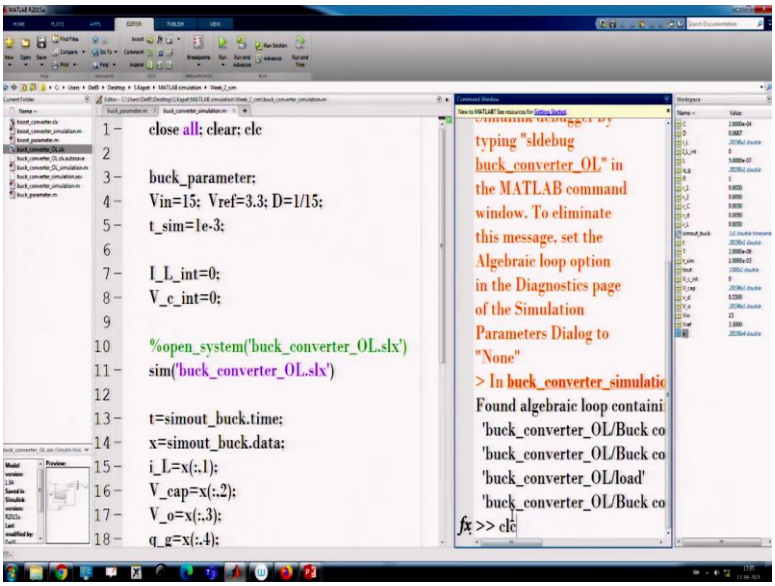
(Refer Slide Time: 30:33)



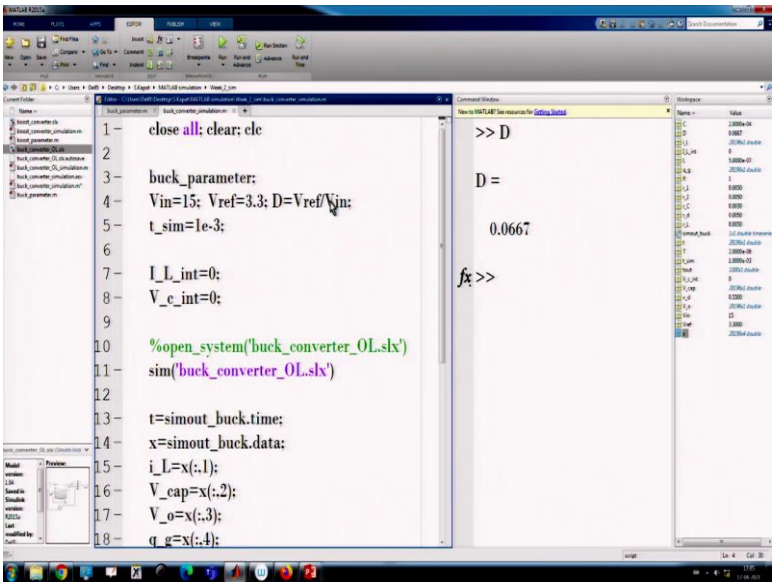
(Refer Slide Time: 30:36)



(Refer Slide Time: 30:51)

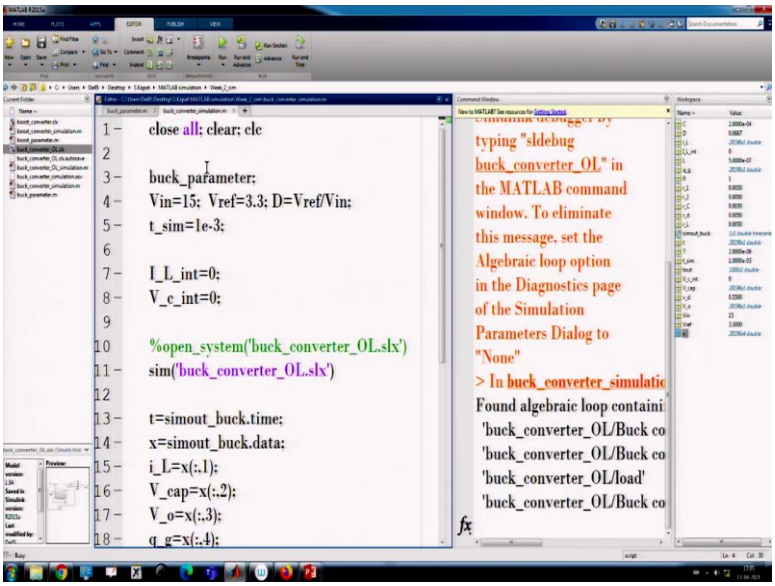


(Refer Slide Time: 30:52)

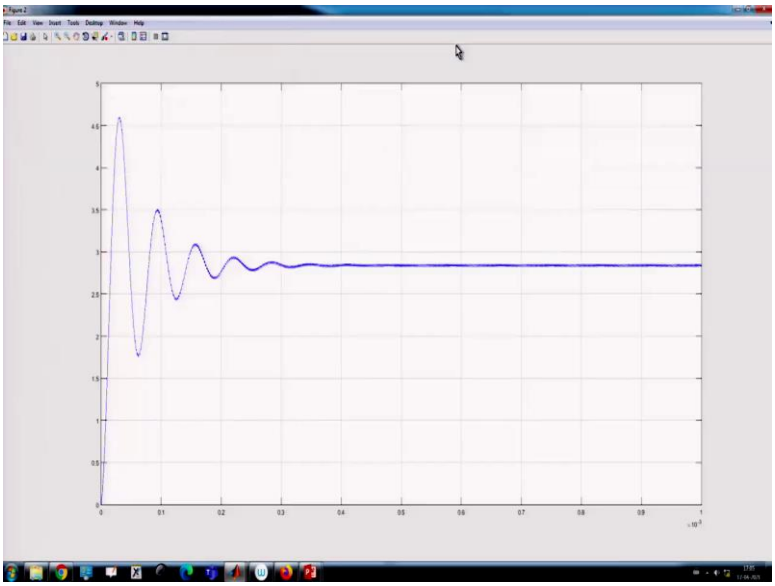


So, in the Simulink D, D should get change. So, ok sorry, we should use Vref by Vin not this, Vref by Vin that is the open loop duty ratio. Now, you should run.

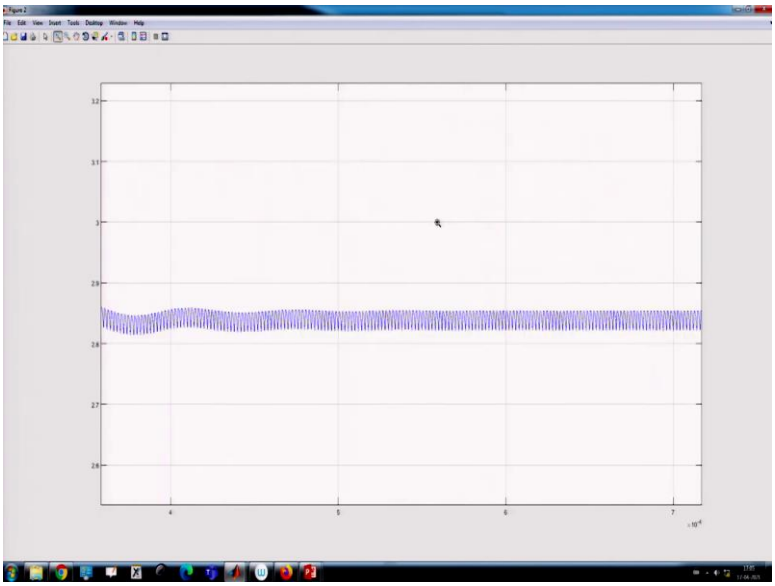
(Refer Slide Time: 31:07)



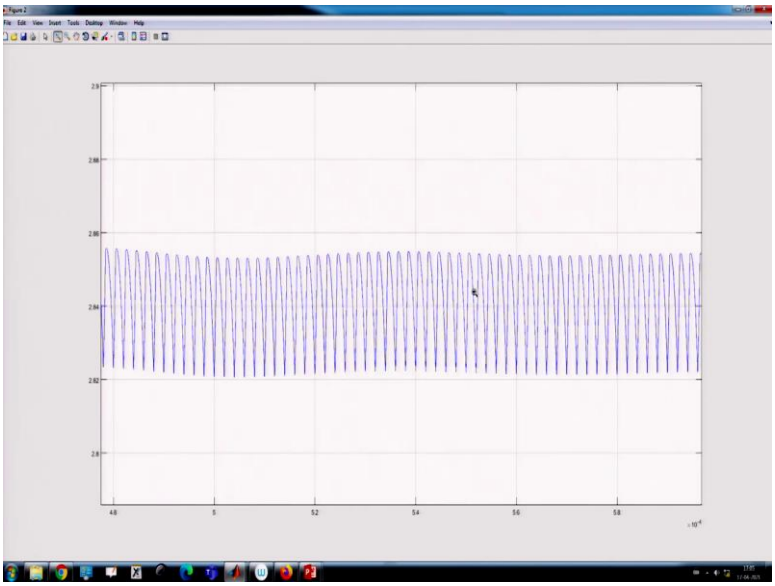
(Refer Slide Time: 31:08)



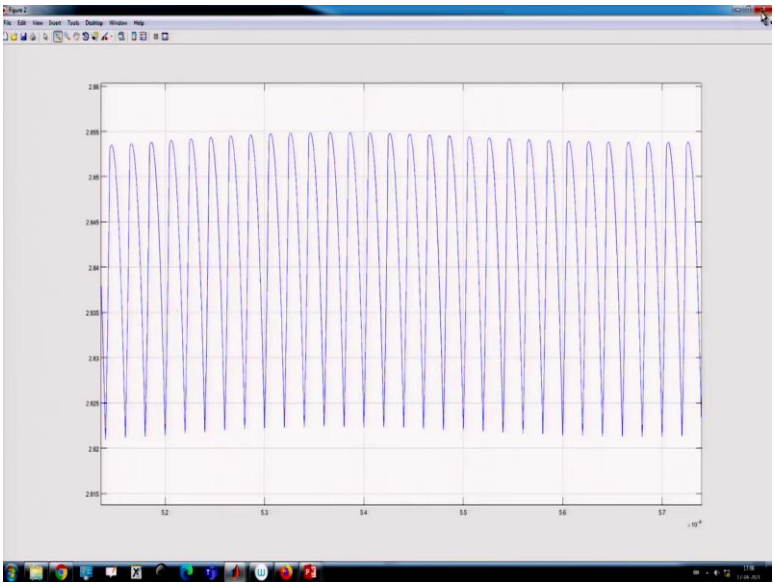
(Refer Slide Time: 31:16)



(Refer Slide Time: 31:18)

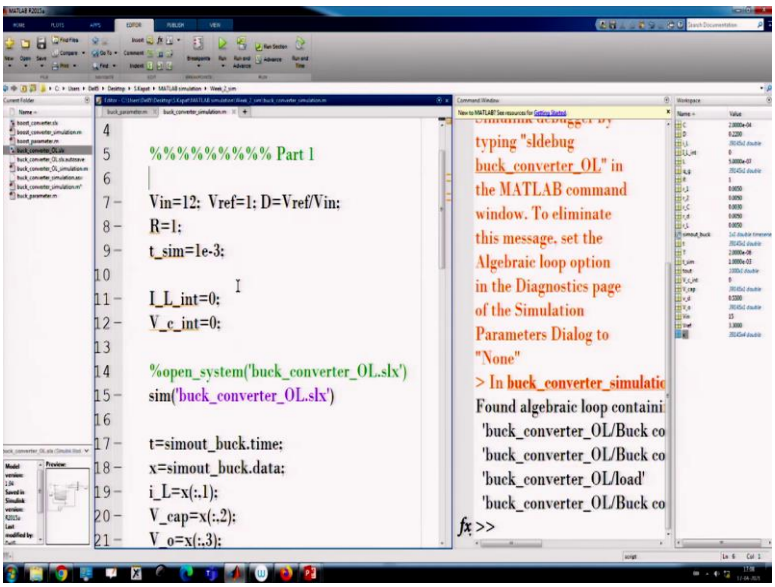


(Refer Slide Time: 31:20)



Now, the output voltage get actually get shifted, but it is not exactly 3.3 because there we have discussed that output voltage difference is because of the drop because we have all considered the non-ideal inductor, capacitor all these. So, inductor DCR drop, diode drop, then diode resistance drop many things will contribute to this and mismatch in the output voltage, but, at least we can run the first thing. So, now, we did it.

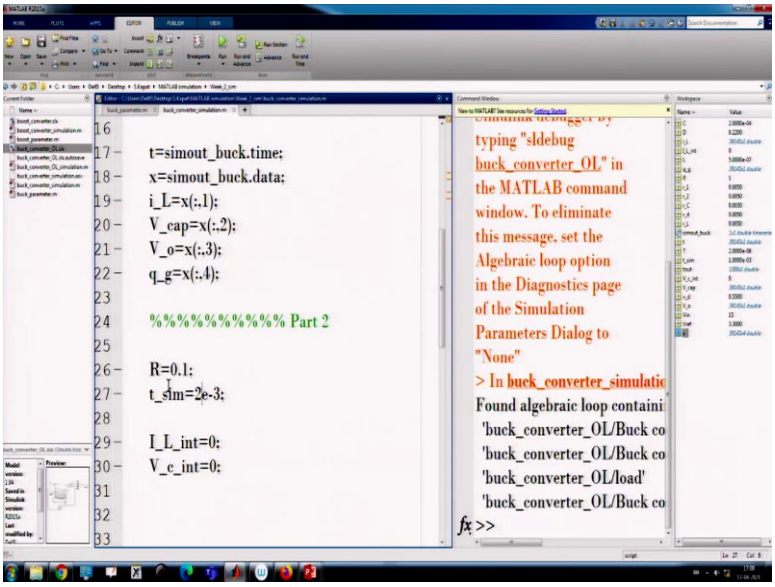
(Refer Slide Time: 32:00)



Now, suppose we want to create another transient case. We do not want to run it now ok; we want to run it later. So, suppose after 1 millisecond, we want to create; we want to

run for another 1 millisecond where suppose we have kept you know R equal to 1; R equal to 1.

(Refer Slide Time: 32:23)



Now, here, we want to set R equal to we want to run for a different R, other parameter remains same so, we are just changing this R value that means, once these data's are stored now, we want to run for another resistance. So, let us say we are going for 1 ohm resistance to 0.1. Let us say it is 1 volt, it is 12 volt. So, all these input voltage is 12 volt, output is desired is 1 volt.

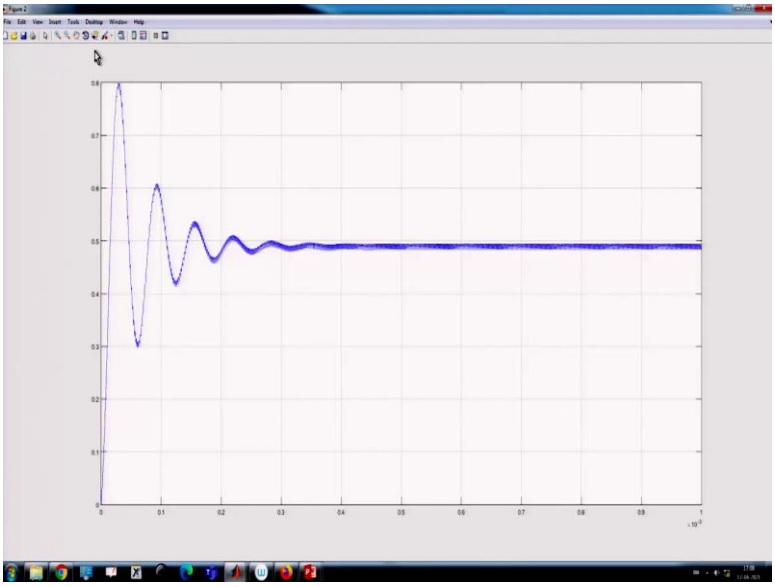
So, we said duty ratio as per open loop, but we can change it and we said the initial load resistance to be 1 ohm, but now we want to make a load transient. So, the load resistance is changed from 1 ohm to 0.1 so that we can get a load current of how much so, that means, load resistance decreases, the current should increase.

But, now, we can run again this simulation. That means, I can just call part 2 of the simulation, part 2 this is my part 1, this file I will call 1's ok, this file I call 1 part 1. If you want to change input voltage fine, if you do not change, then you do not have to write it because it is already stored, but we are changing these values part 2.

So, let us say we are running for 2 millisecond, but one thing we will notice that now, the inductor current we are running for the next we already run for 1 millisecond. We have

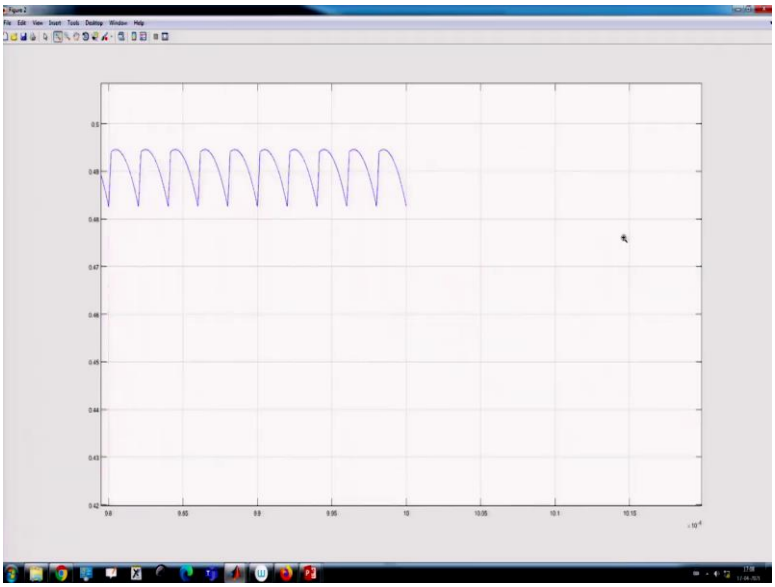
to run for 2 another 2 millisecond, we cannot run from the same initial condition 0 because the already system has reached to certain level ok. So, let me let us draw this.

(Refer Slide Time: 34:05)

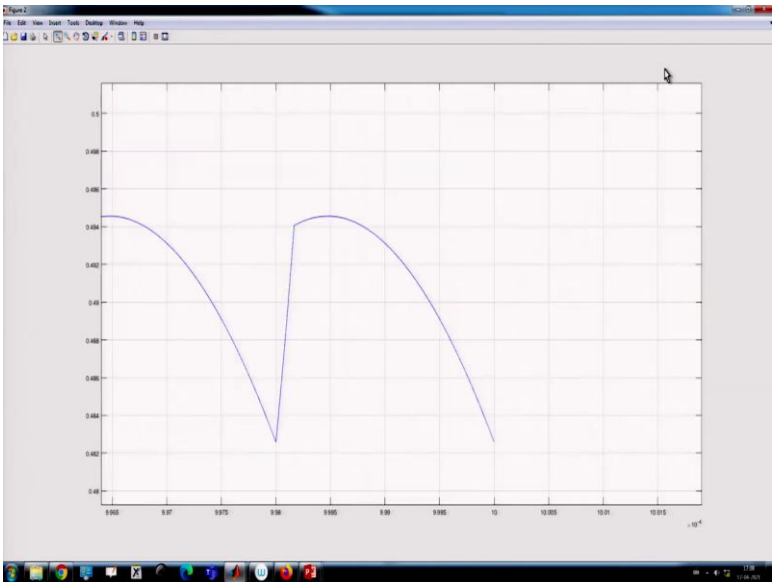


So, let us say the output voltage reached here some value.

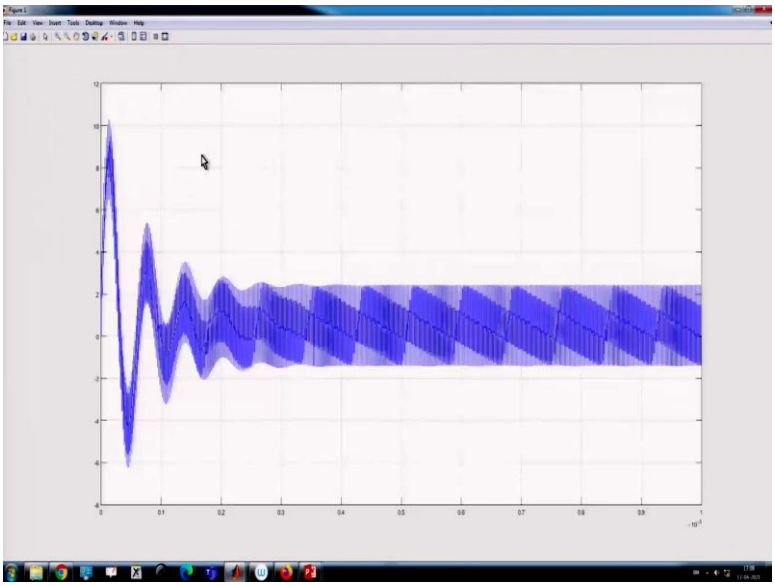
(Refer Slide Time: 34:11)



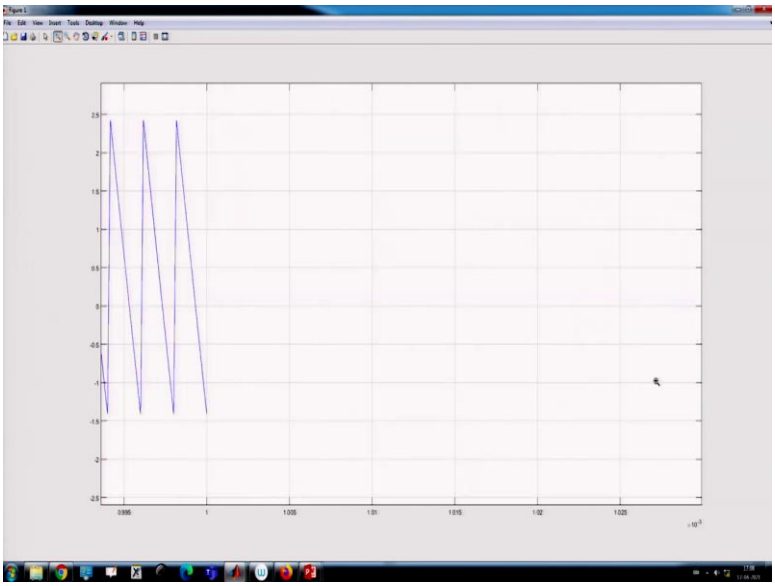
(Refer Slide Time: 34:13)



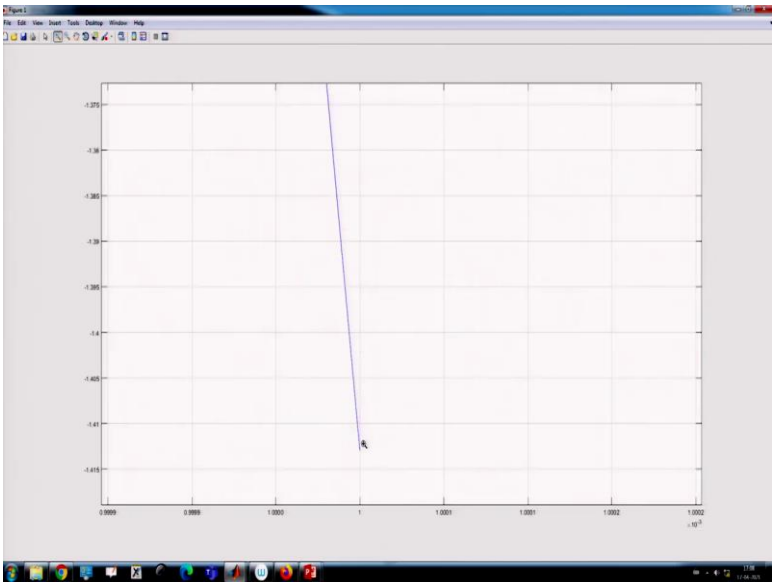
(Refer Slide Time: 34:14)



(Refer Slide Time: 34:20)

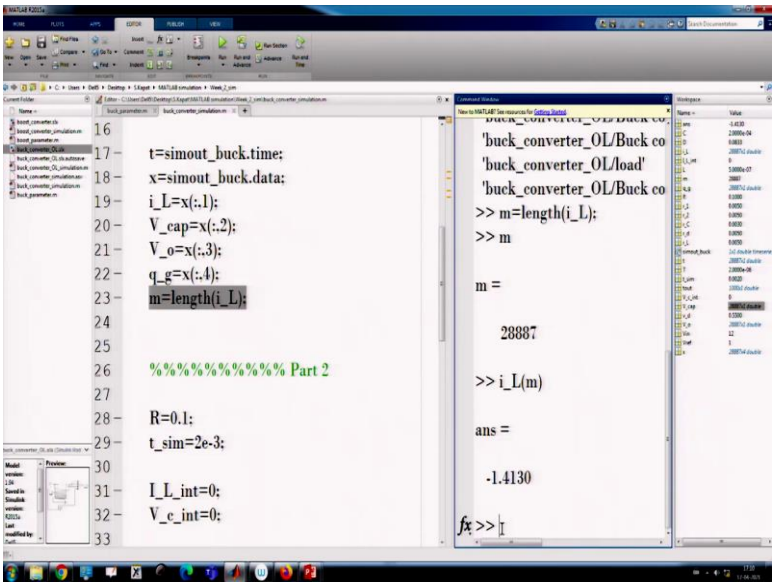


(Refer Slide Time: 34:22)



So, we have to take or inductor current has reached some value whether it is positive, negative, but it is at the final value, it is not 0. So, we have to start from this point rather than another 0 point. How to do that? That means, we have to take the final value of the previous simulation as the initial value of the next simulation.

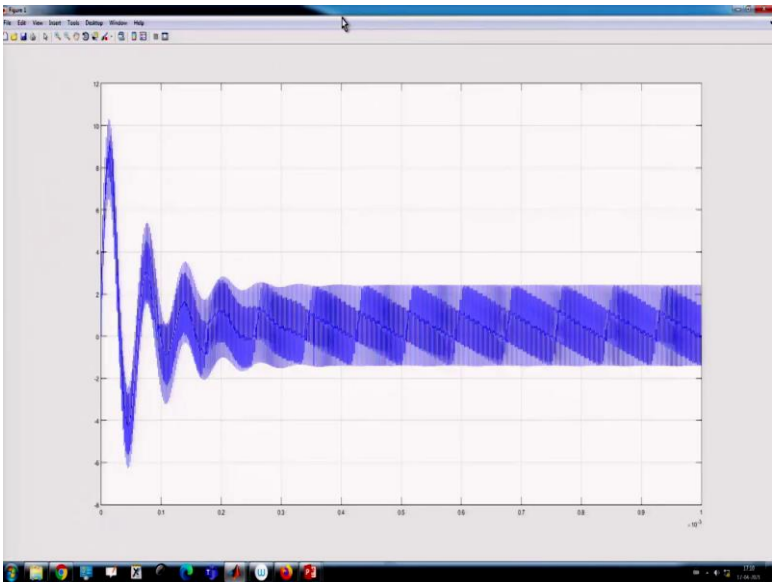
(Refer Slide Time: 34:57)



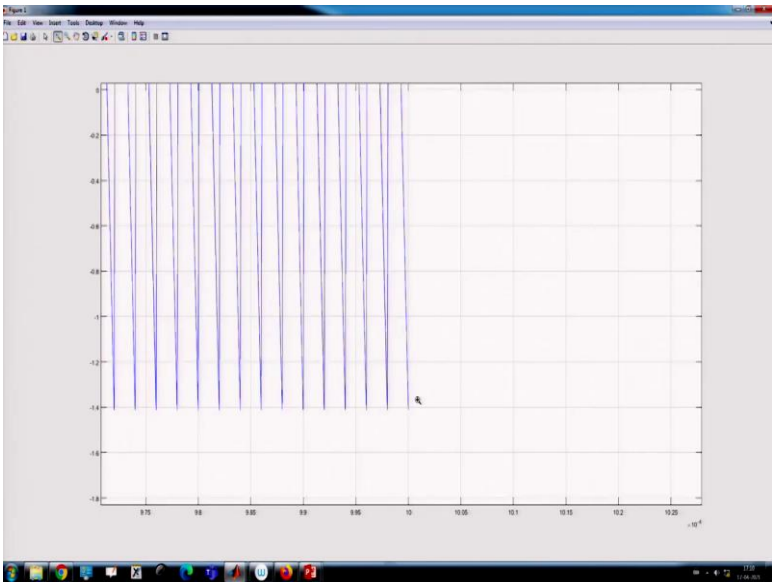
So, what is the final value of the previous simulation? So, we need to identify what is the vector length of these any of all these vectors are identical length and it is clear that if you look at this; it is like a 28887, but that can vary. So, I can simply write m which is the length of any of the signal I can use i L for example. If I execute this, it will show the length of this vector and what is my m? Like a 28887.

So, we have to use this data as the initial condition that means, what is the initial condition for the next simulation? It is the final condition for this. So, if I search for i underscore L ok m; m, then what is this value? Minus 1.413 and if you look at this waveform sorry actually, what was where is the scope result? If we just run it you will find the final value of this inductor current.

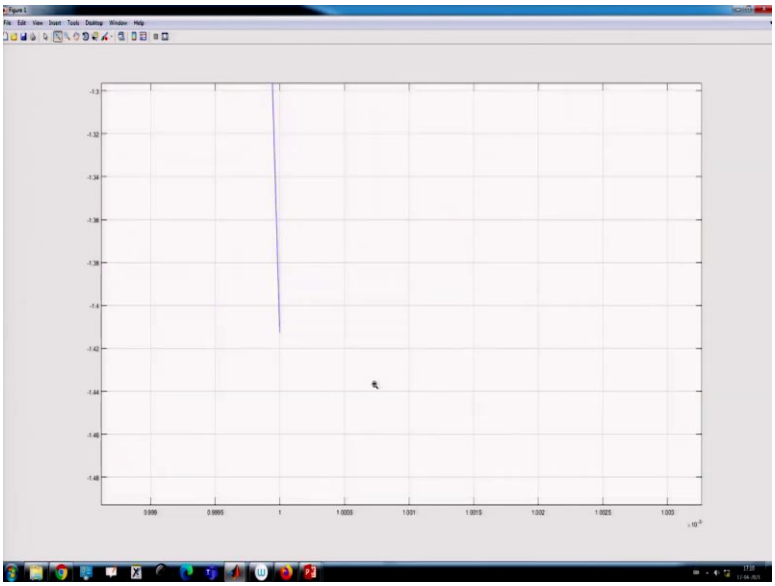
(Refer Slide Time: 35:49)



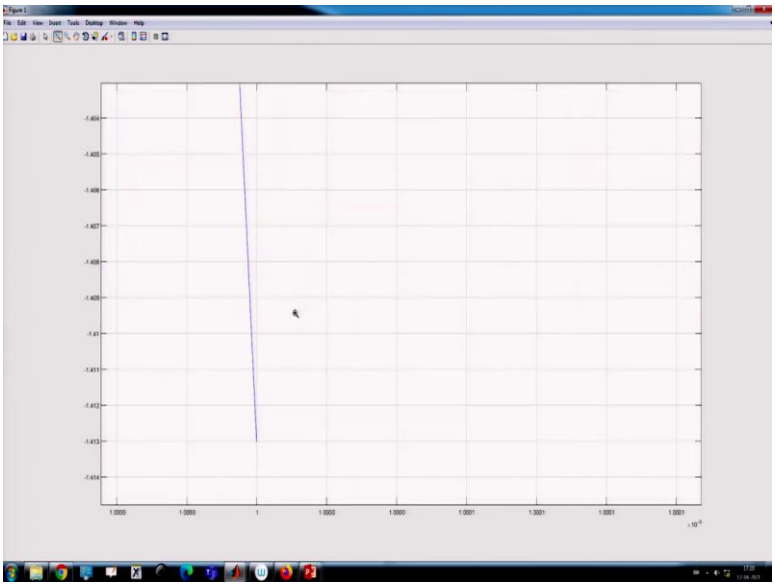
(Refer Slide Time: 35:54)



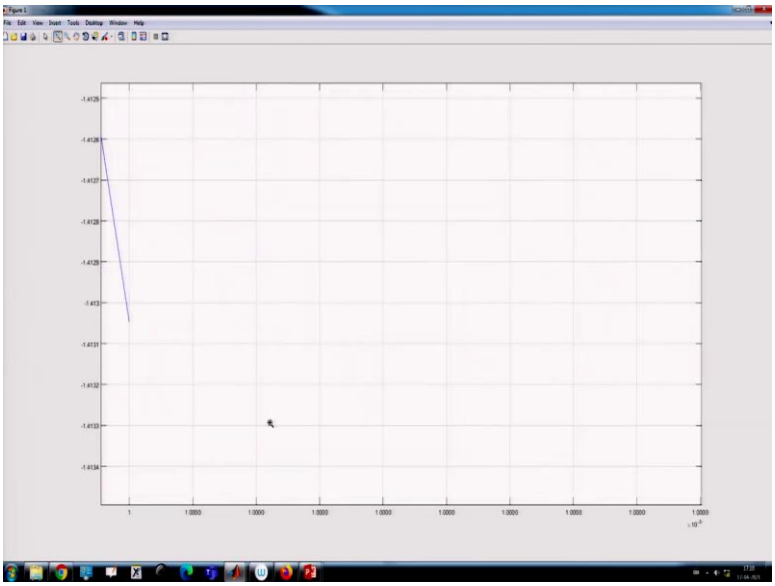
(Refer Slide Time: 35:55)



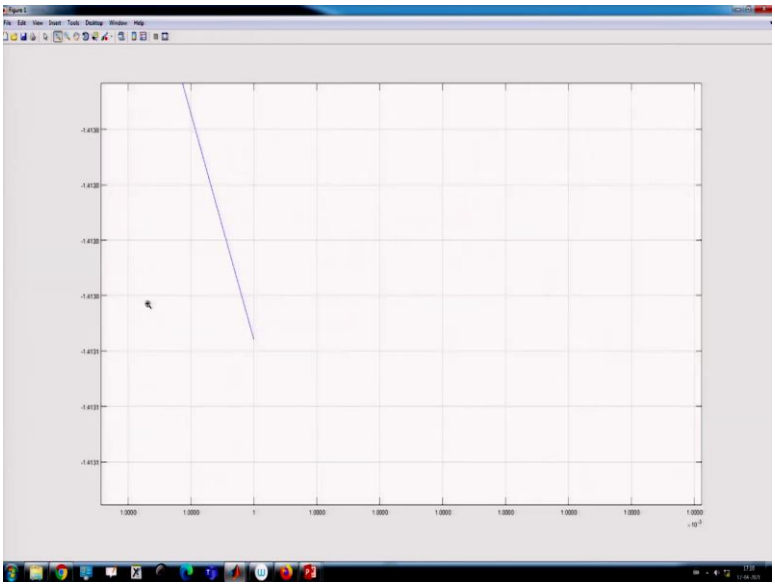
(Refer Slide Time: 35:57)



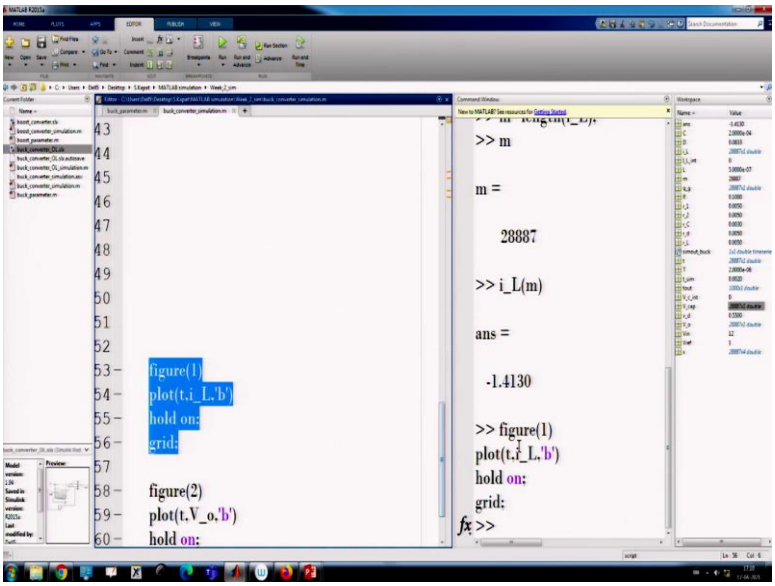
(Refer Slide Time: 35:59)



(Refer Slide Time: 36:02)

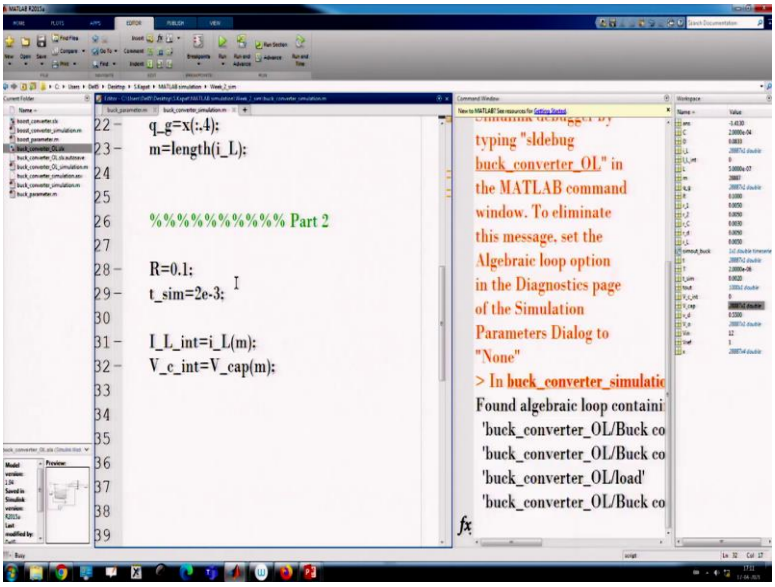


(Refer Slide Time: 36:05)



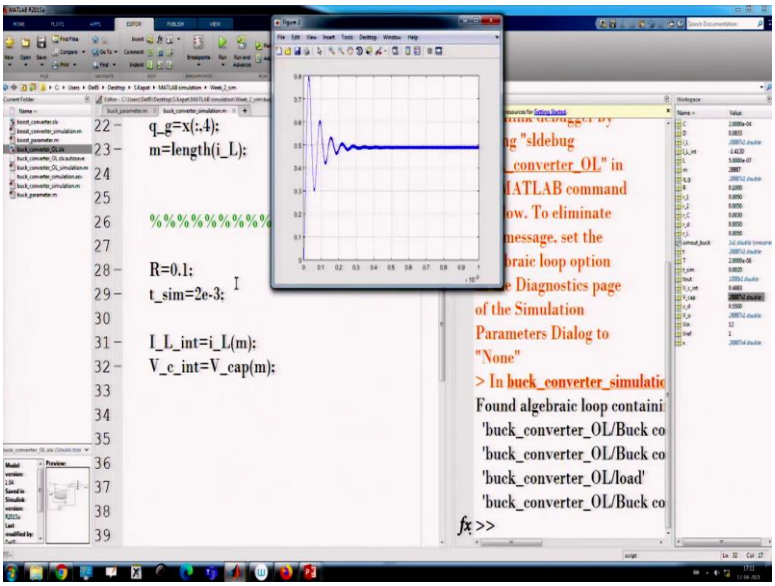
If you go and go, this is what exactly it is showing that minus 1.413 so, that is showing here minus 1 point so, that is my final value.

(Refer Slide Time: 36:24)

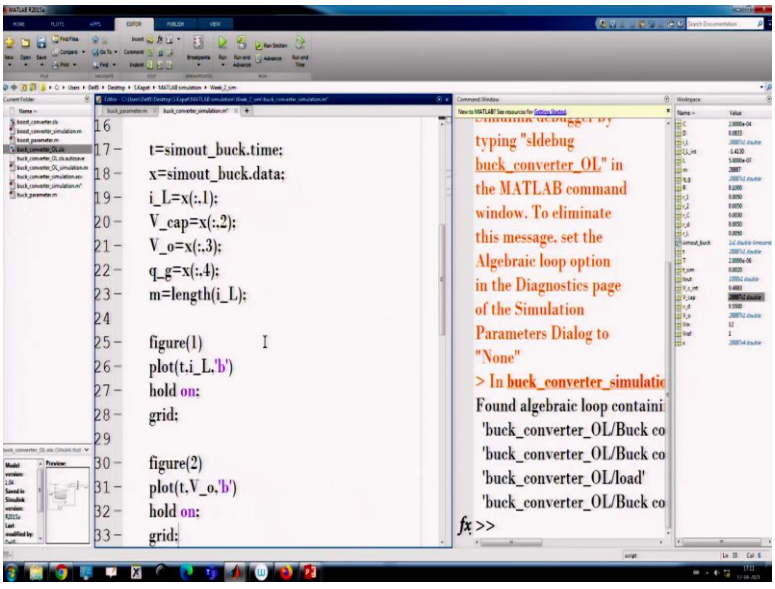


That means, first, that means we need to identify the final value. So, this initial condition will be my i L comma m and this initial condition will be my V cap comma m ok. So, now, you are ready to go.

(Refer Slide Time: 36:37)

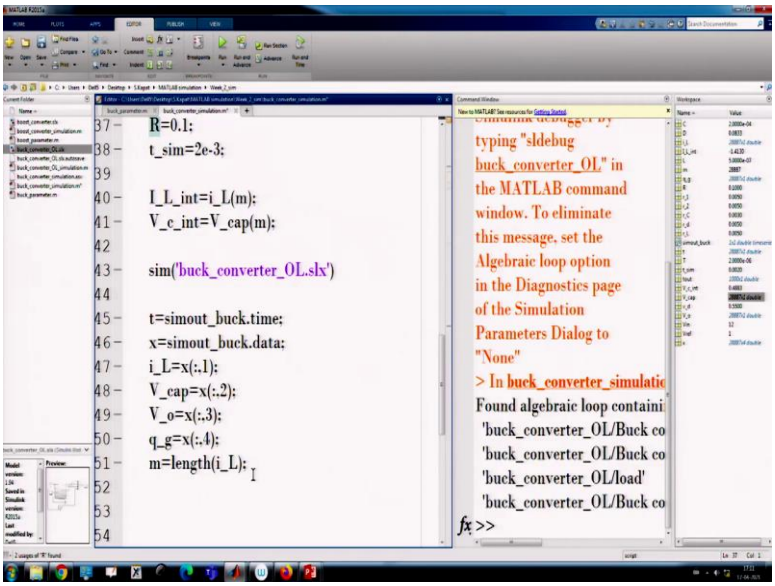


(Refer Slide Time: 36:58)



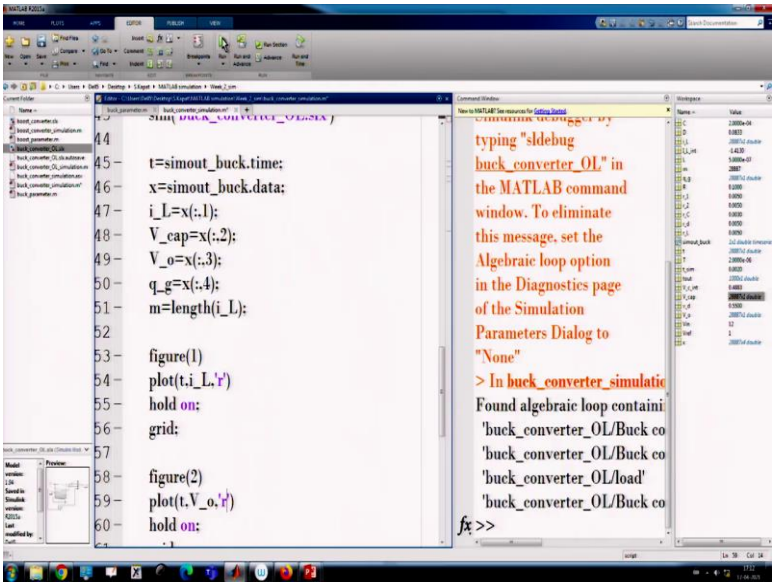
Now, I want to run you know I want to hold on this waveform. So, this simulation as if is created ok. So, part 1 let us say we are running this output to voltages where it is under part 1 that part 1, I am plotting all these holding the value. Now, what I am doing? I am again calling this function sim function and doing this exercise ok; doing this exercise sim.

(Refer Slide Time: 37:14)



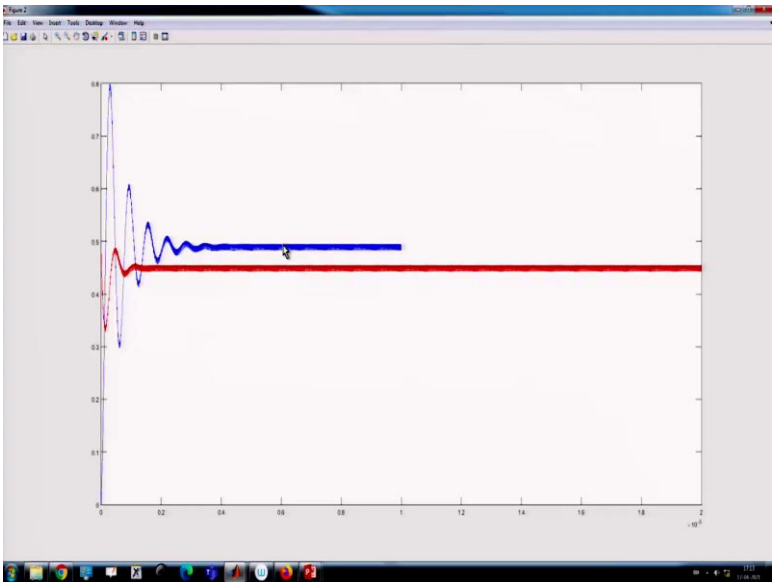
Now, for this, I set the initial condition; I have only changed the load resistance; I am calling everything and again; I am I can plot this. Now, all these data will be overwritten.

(Refer Slide Time: 37:33)

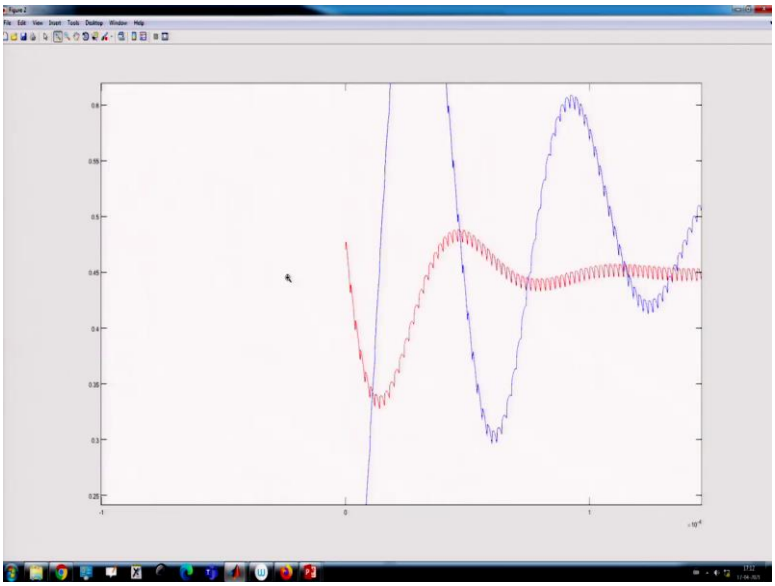


And now, I plot is a red one, red one. So, let us run the simulation.

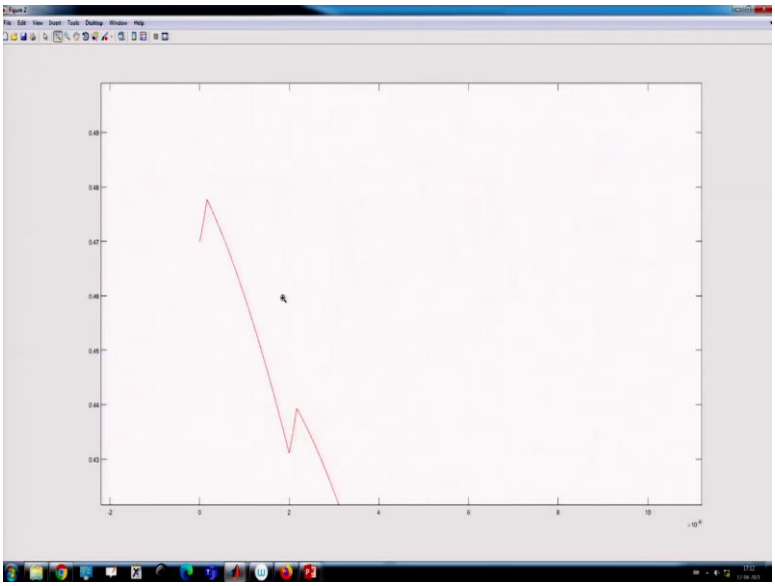
(Refer Slide Time: 37:38)



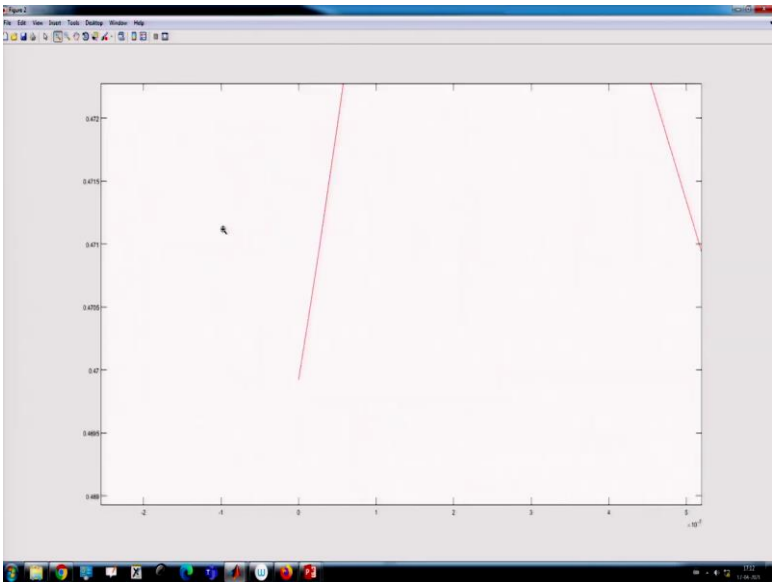
(Refer Slide Time: 37:44)



(Refer Slide Time: 37:46)

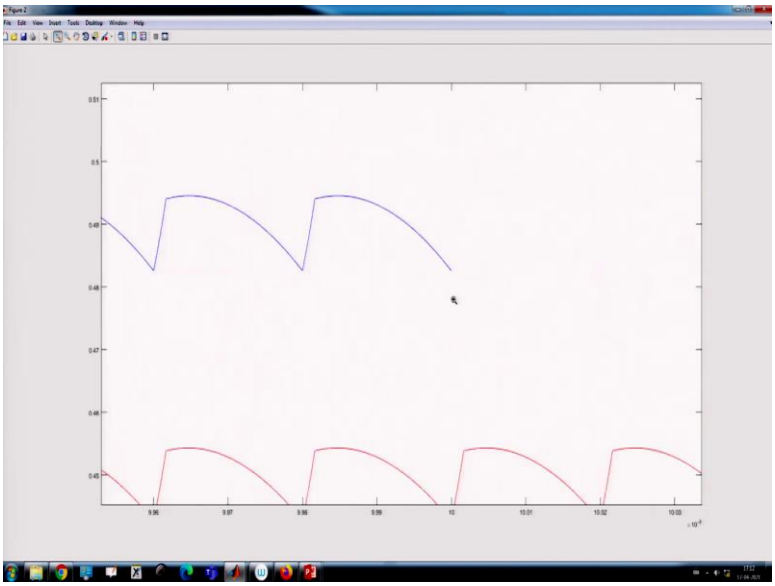


(Refer Slide Time: 37:48)

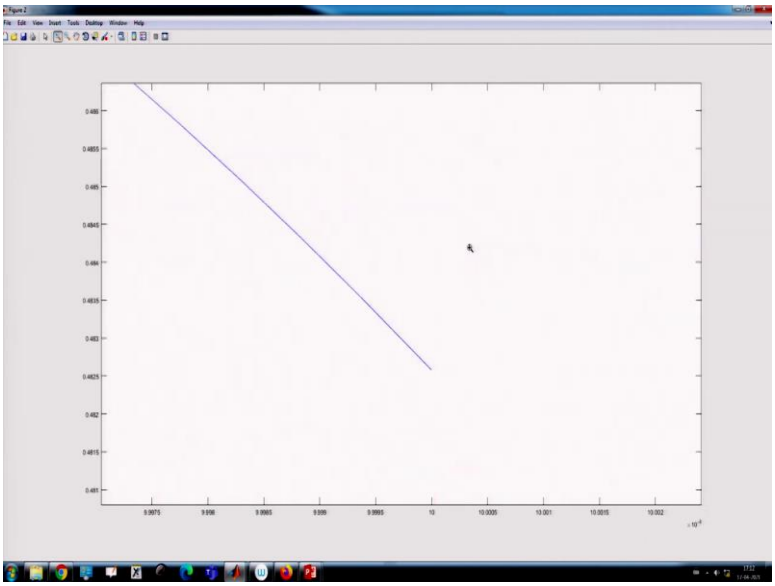


If you run, you will find the initial value of this is updated accordingly. What is the initial value? It is around 0.47.

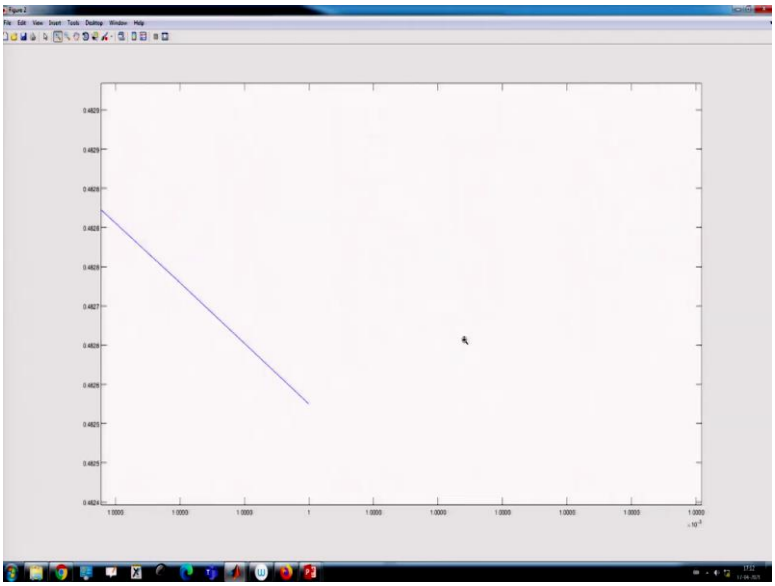
(Refer Slide Time: 37:55)



(Refer Slide Time: 37:57)

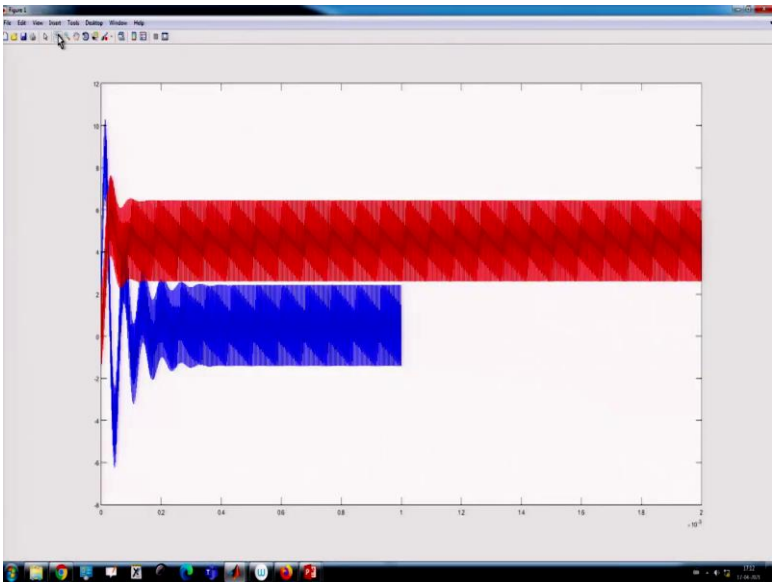


(Refer Slide Time: 38:00)

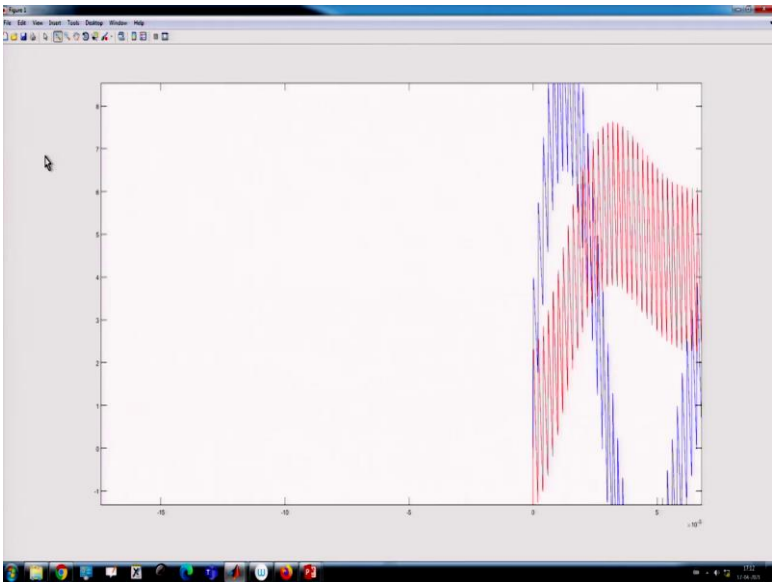


And what is the final value here? It is around sorry, what I am showing sorry.

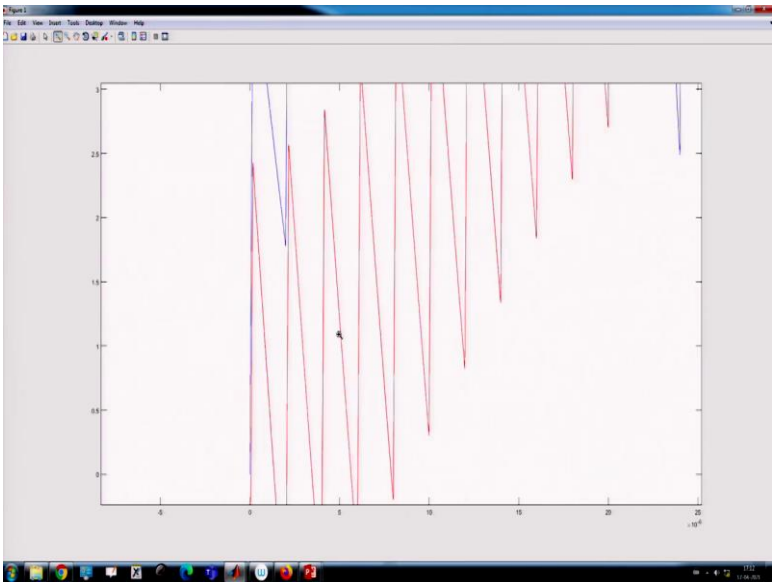
(Refer Slide Time: 38:05)



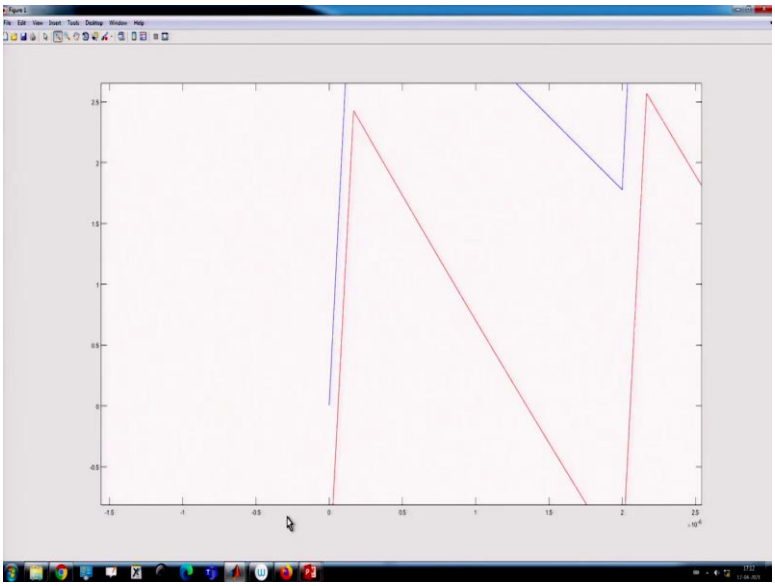
(Refer Slide Time: 38:14)



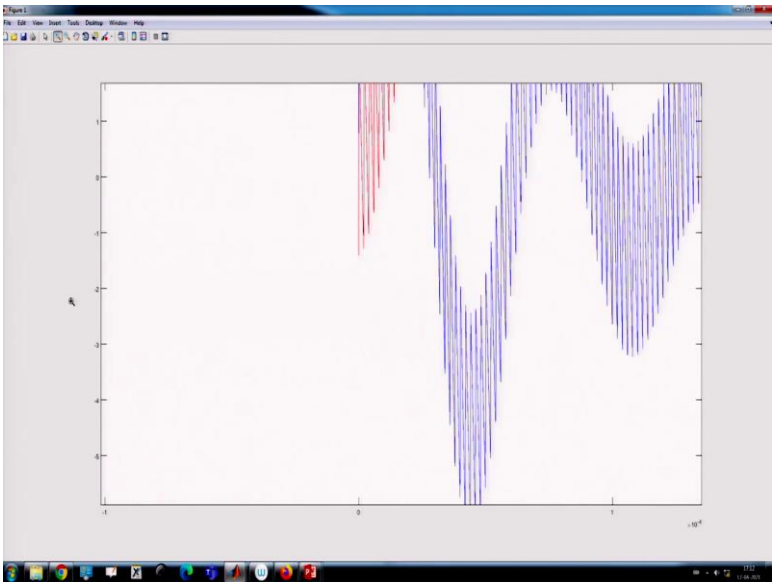
(Refer Slide Time: 38:16)



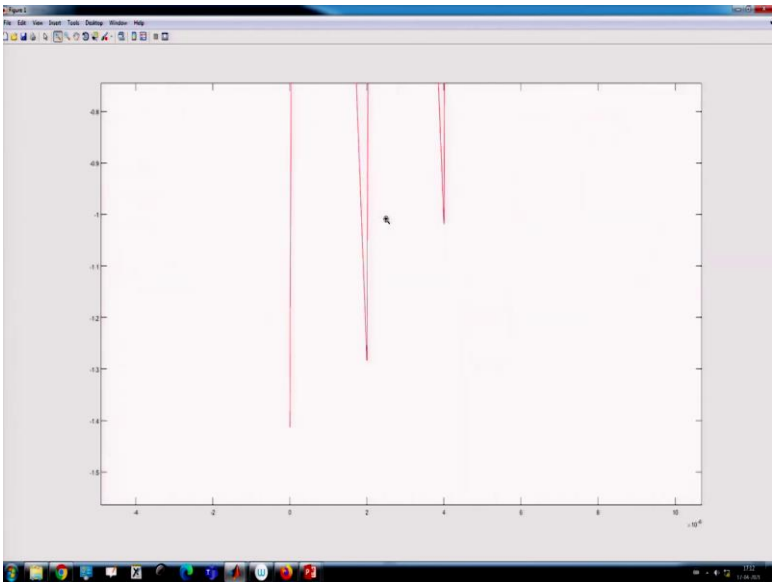
(Refer Slide Time: 38:18)



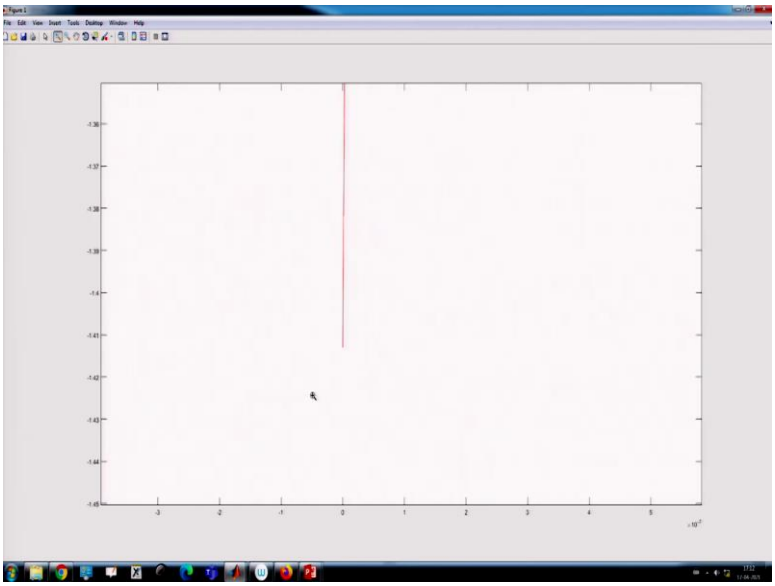
(Refer Slide Time: 38:22)



(Refer Slide Time: 38:24)

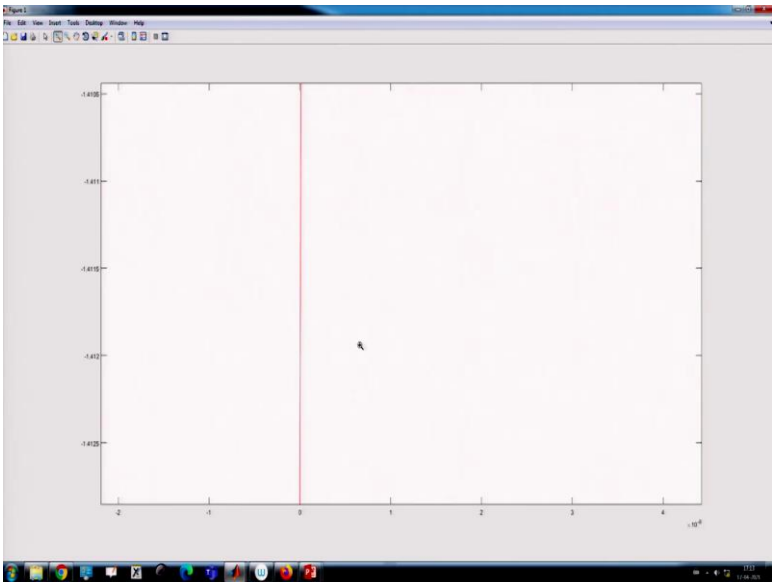


(Refer Slide Time: 38:26)

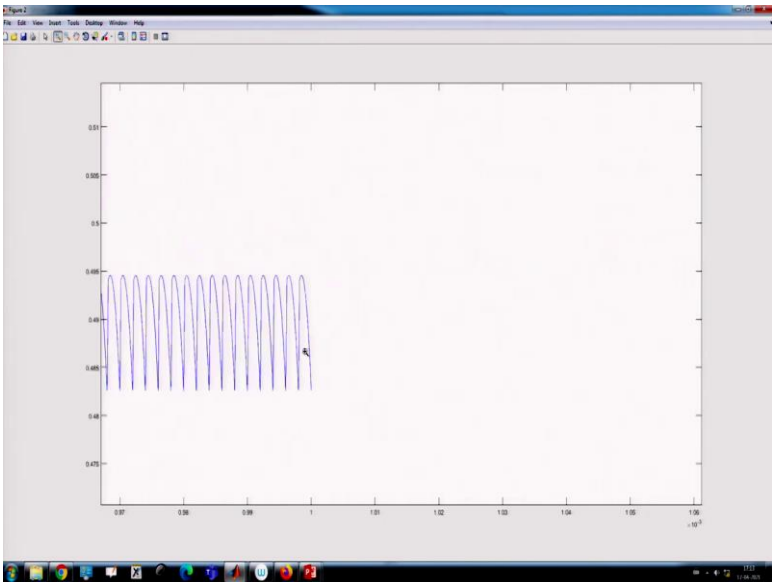


So, the final condition here, that means, this is my initial condition yeah, initial condition so, red one if you see the final initial condition that we discuss minus 41 that value and so, it is fine.

(Refer Slide Time: 38:29)

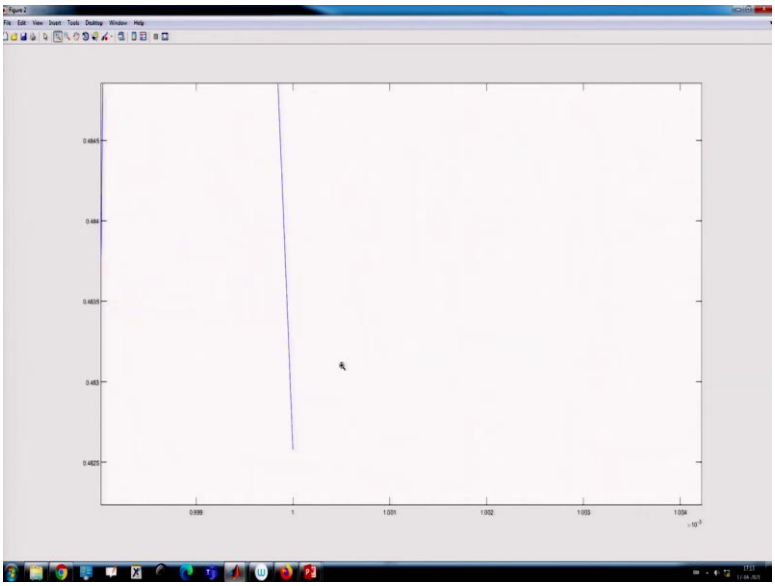


(Refer Slide Time: 38:41)

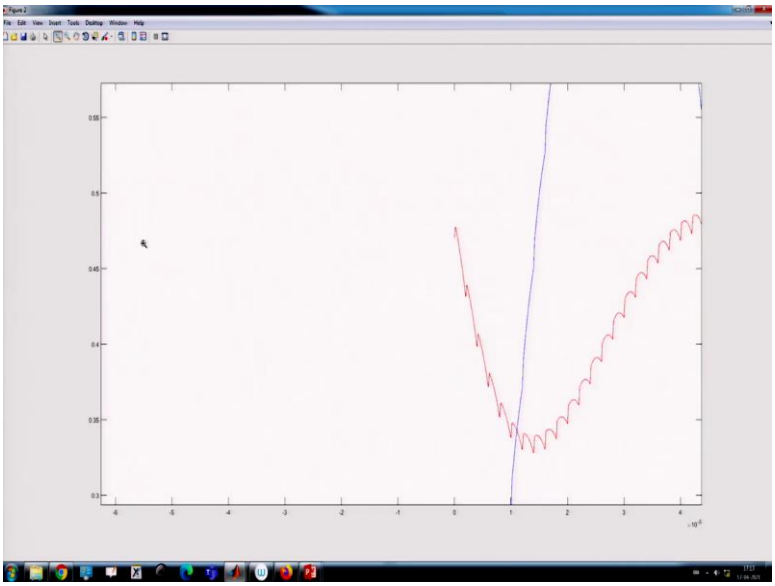


So, that means, we can see some drop that means, if you run again so, you will check there is a some drop in the output voltage. Output voltage final value is around 0.4825.

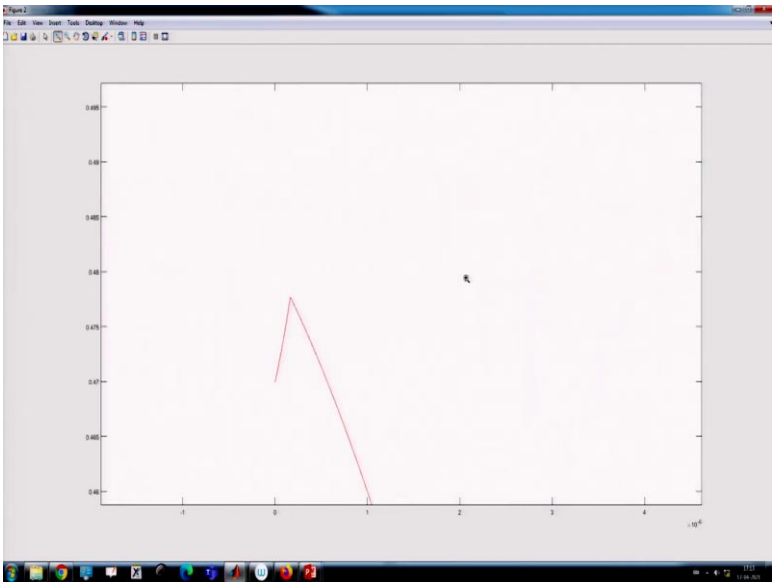
(Refer Slide Time: 38:43)



(Refer Slide Time: 38:49)

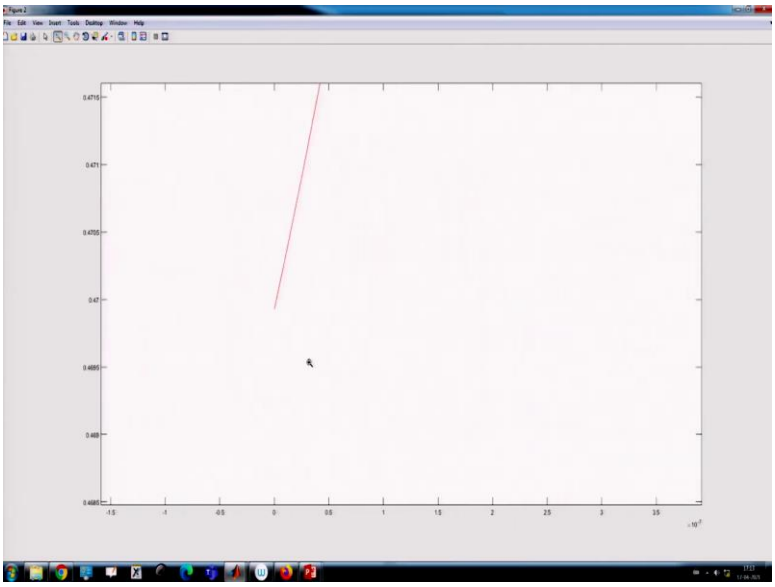


(Refer Slide Time: 38:50)



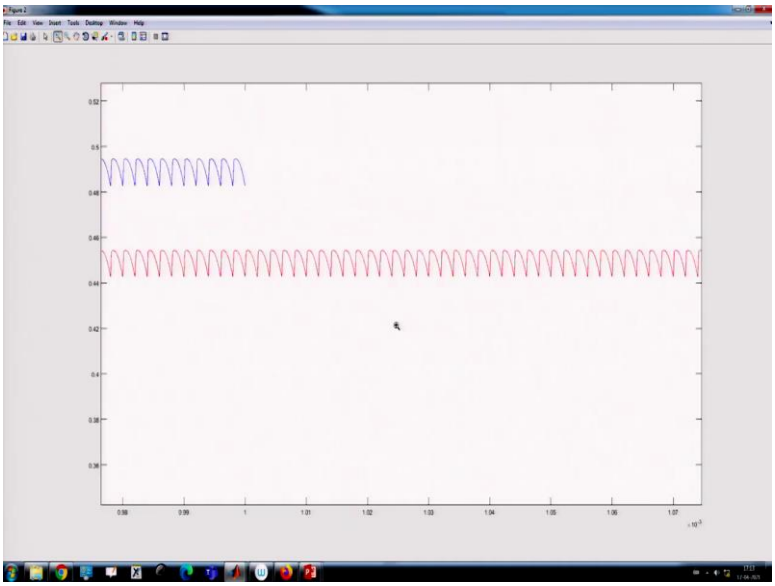
Whereas, the initial value of the next simulation is something different.

(Refer Slide Time: 38:52)



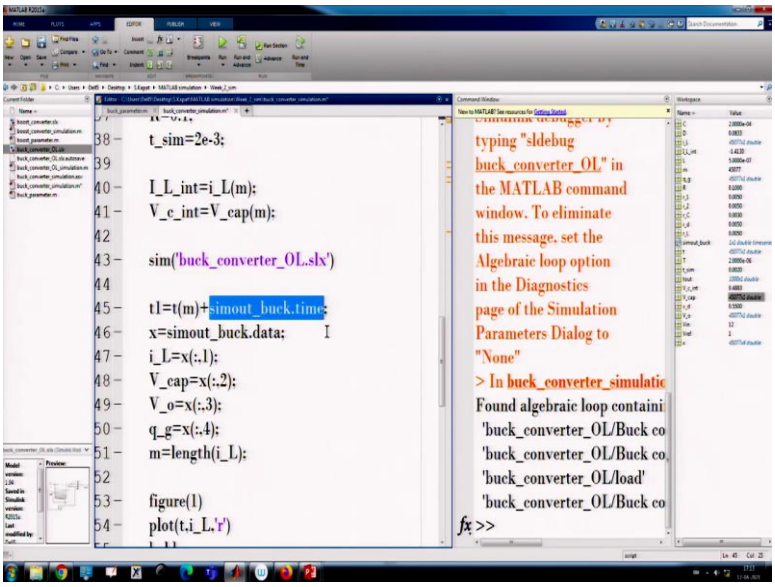
Another thing you will notice the time shift as if there again the simulation is running from 0 to t sim; 0 to t sim.

(Refer Slide Time: 39:03)



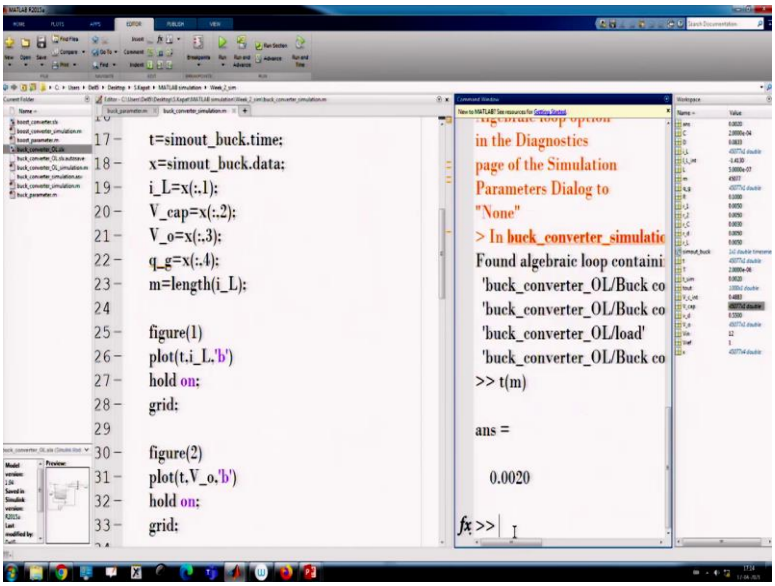
So, we need to shift it that means, we have to shift to this access.

(Refer Slide Time: 39:15)



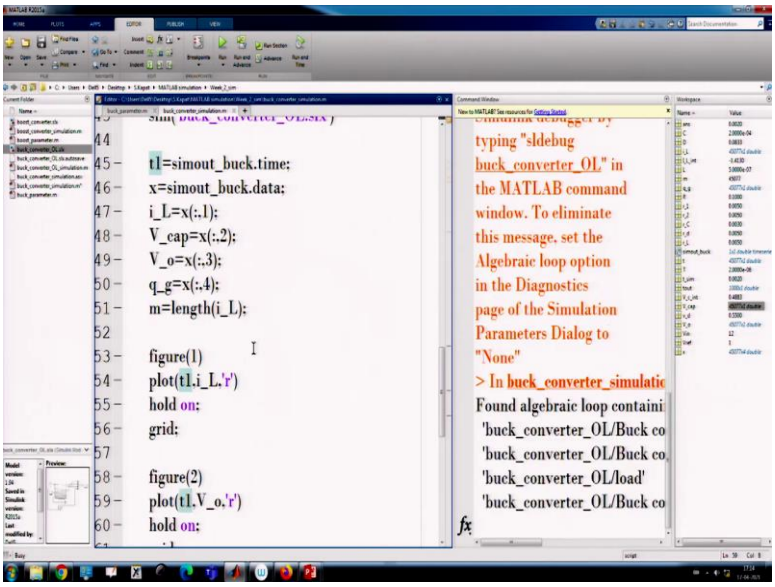
How to do that? So, in order to shift that now, we have to create the present time, which is like t1, it should be the previous time which was my t m vector plus why? Because this is the vector, time vector, but this time vector should we are adding an offset value so, the whole vector will be shifted by that time and what is the offset value? That is the final time value in the previous simulation that means t.

(Refer Slide Time: 39:41)



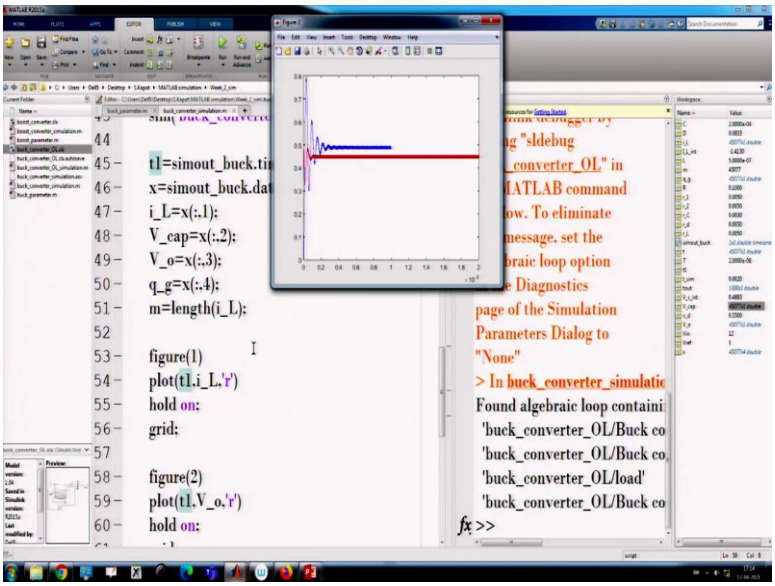
So, if we write t m so, final time vector because they have updated.

(Refer Slide Time: 39:55)

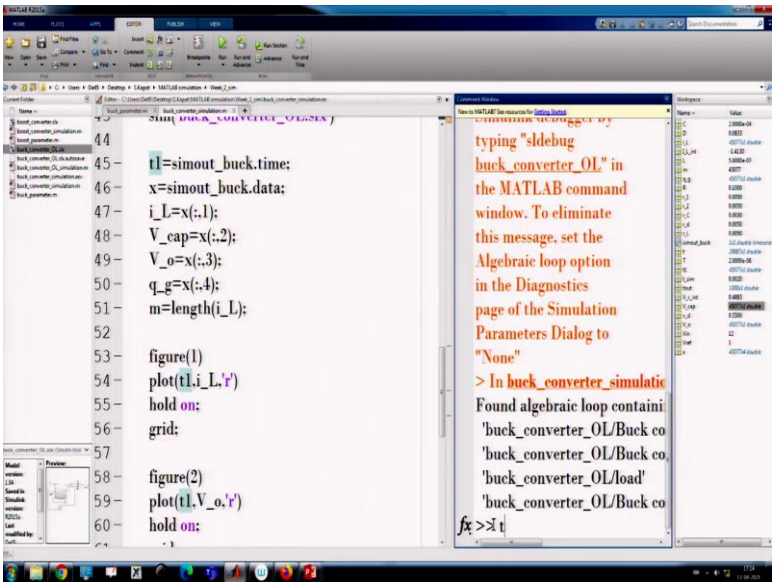


So, let us do one thing in order to avoid any confusion so, we will use this t1, t1, t1 ok, we will run it again everything is fine.

(Refer Slide Time: 40:00).

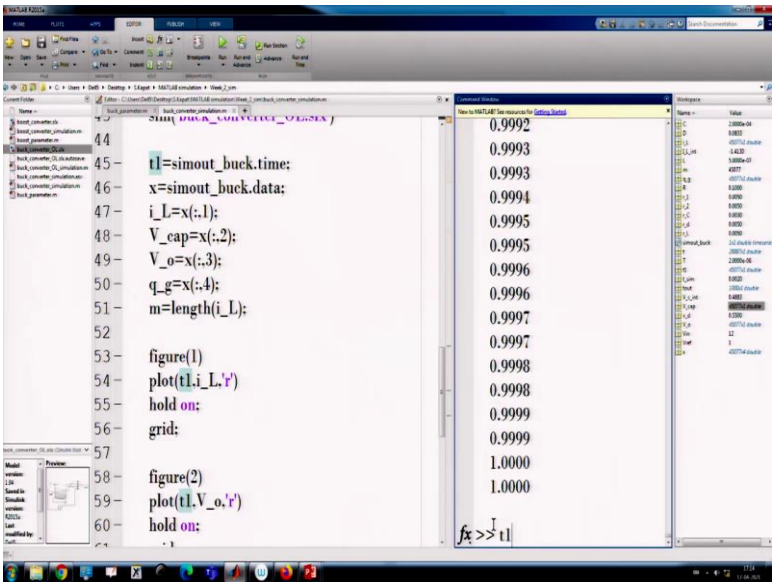


(Refer Slide Time: 40:02)



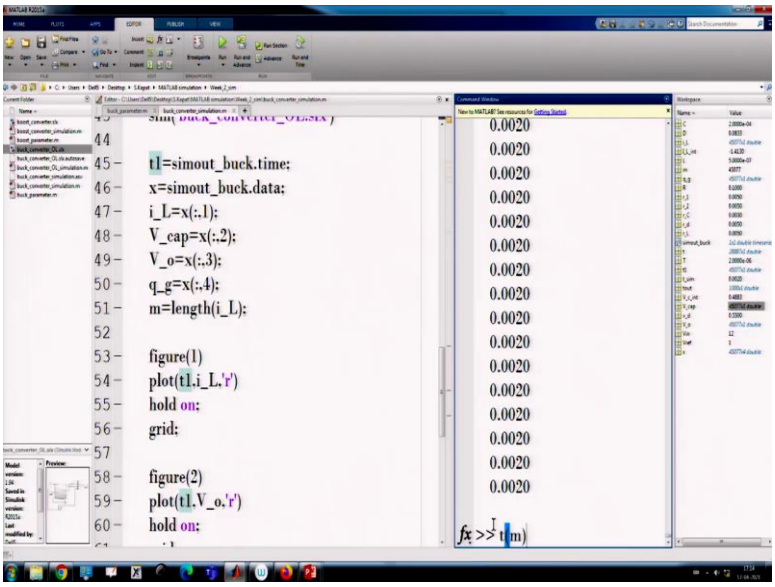
Now, what is t? t is this.

(Refer Slide Time: 40:03)



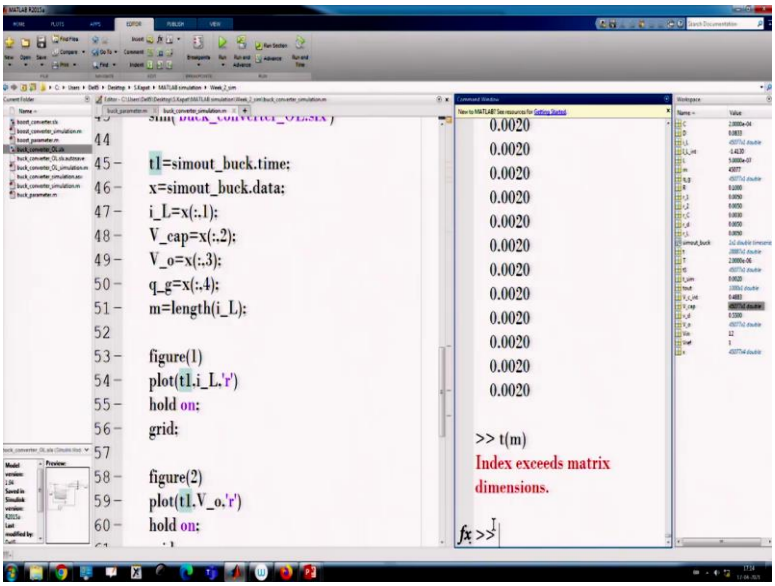
What is t 1? t 1 is this.

(Refer Slide Time: 40:05)



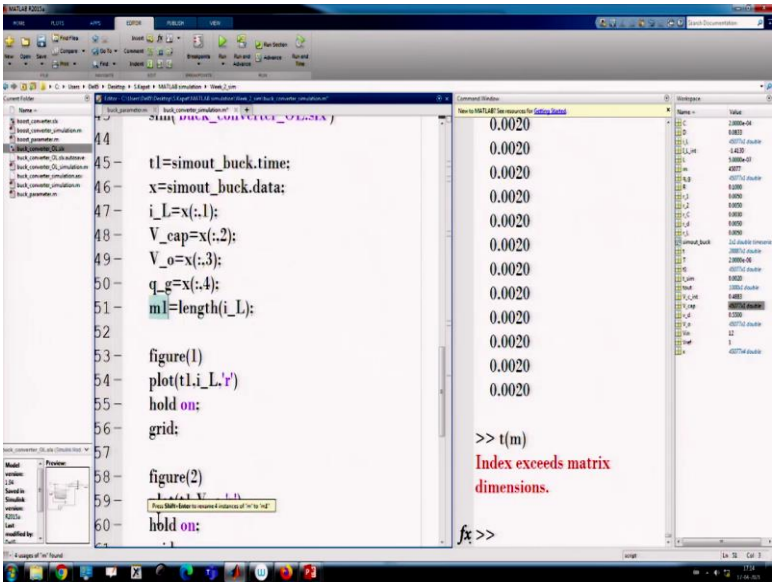
What is t of m this?

(Refer Slide Time: 40:09)



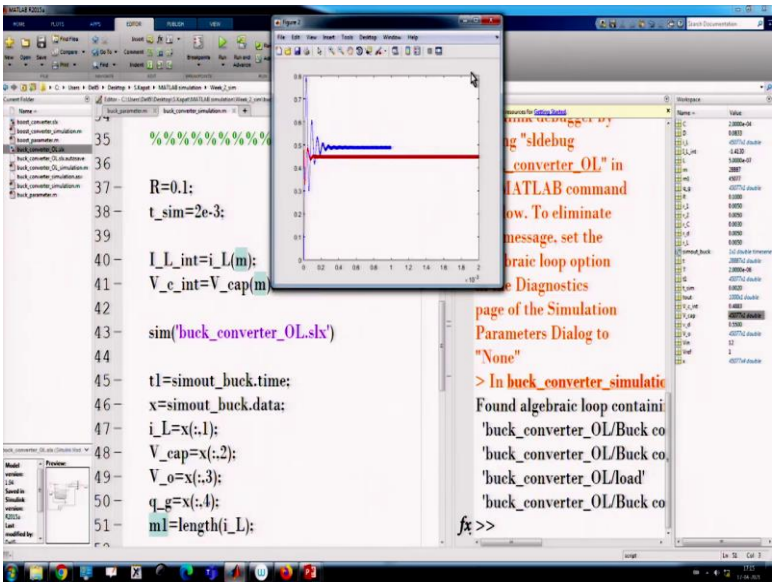
m actually does not exist because it has updated the length, this length is no longer, this is the earlier length.

(Refer Slide Time: 40:24)

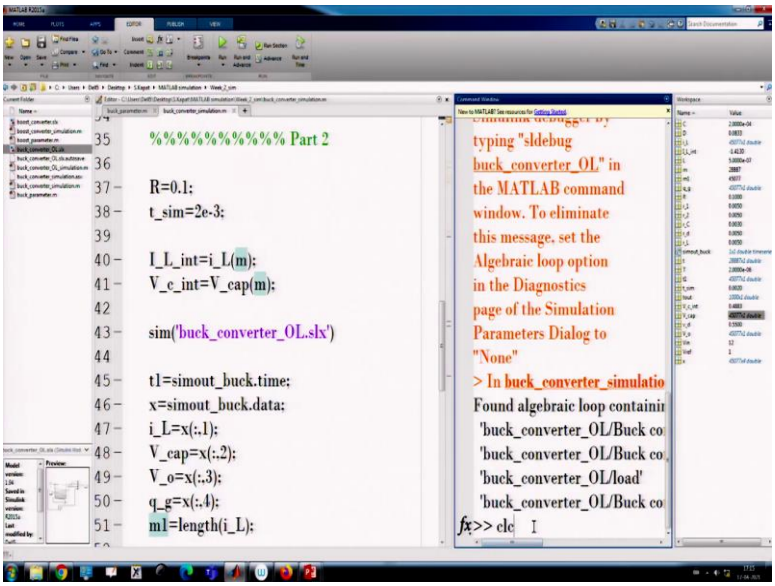


M this is let us say m1 ok.

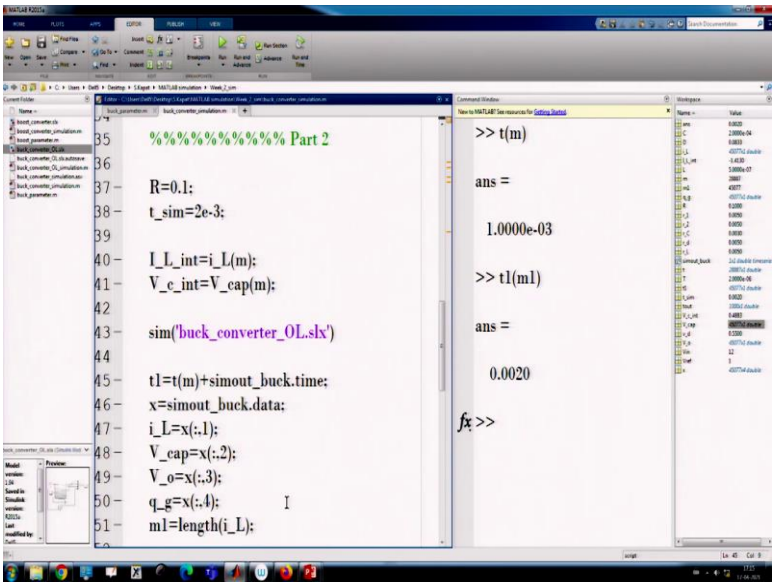
(Refer Slide Time: 40:31)



(Refer Slide Time: 40:35)

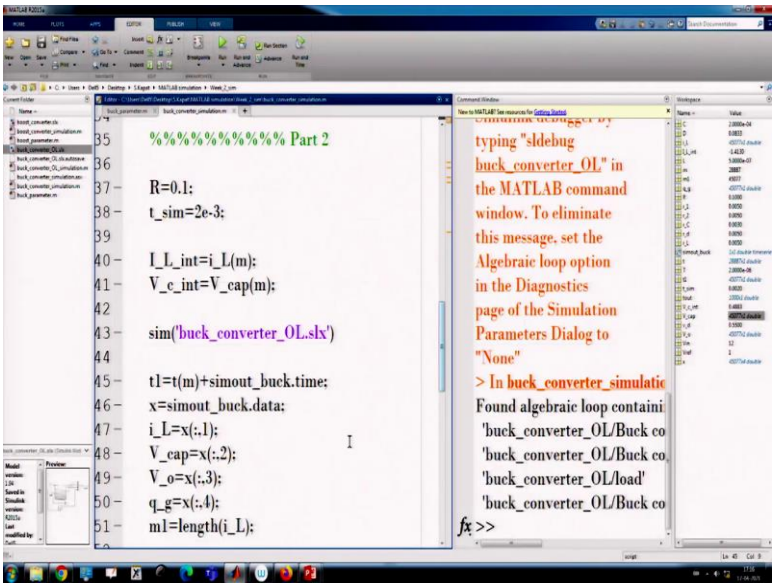


(Refer Slide Time: 40:36)



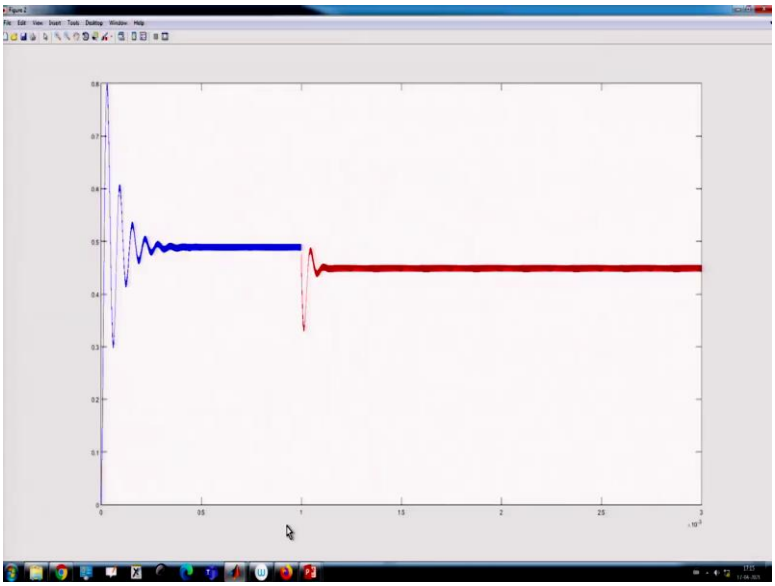
Now, we want to see what is my t of m? What is my t1 of m1? Yeah so, different value. So, now what we have d do? We have to add this offset t m ok t m plus, we have to add this offset ok.

(Refer Slide Time: 41:00)

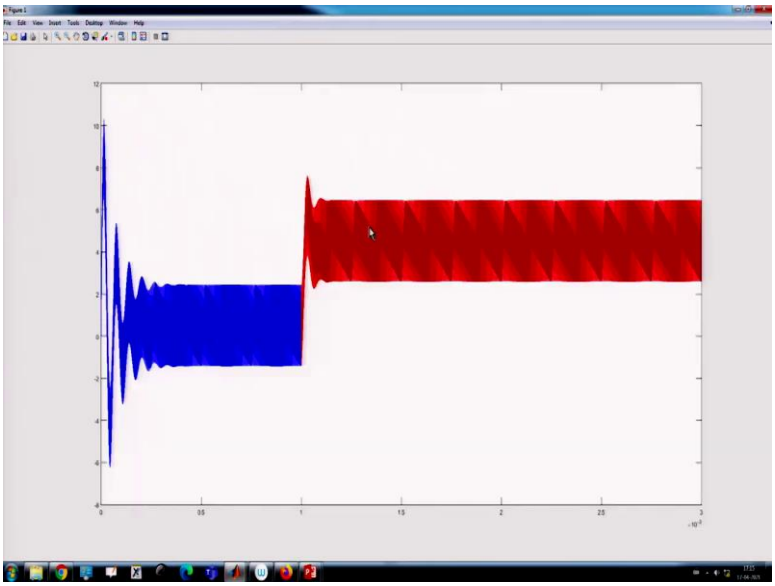


Now, we if we add this offset, run it.

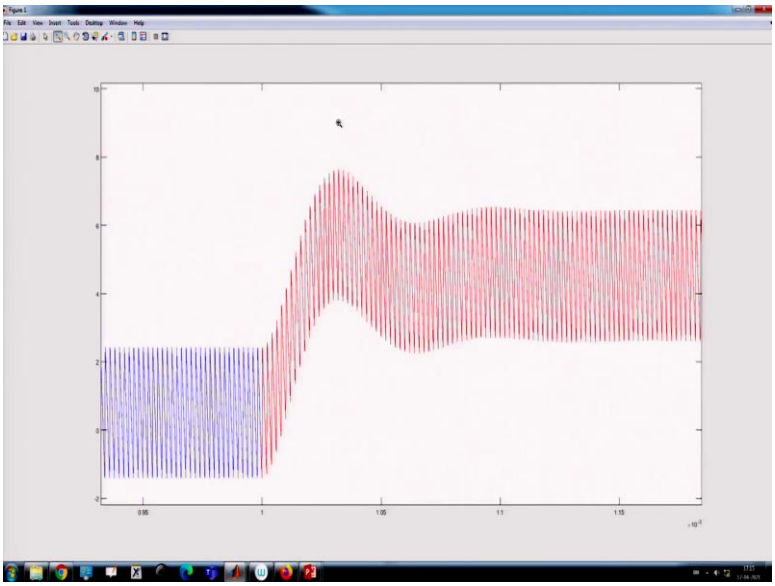
(Refer Slide Time: 41:02)



(Refer Slide Time: 41:09)

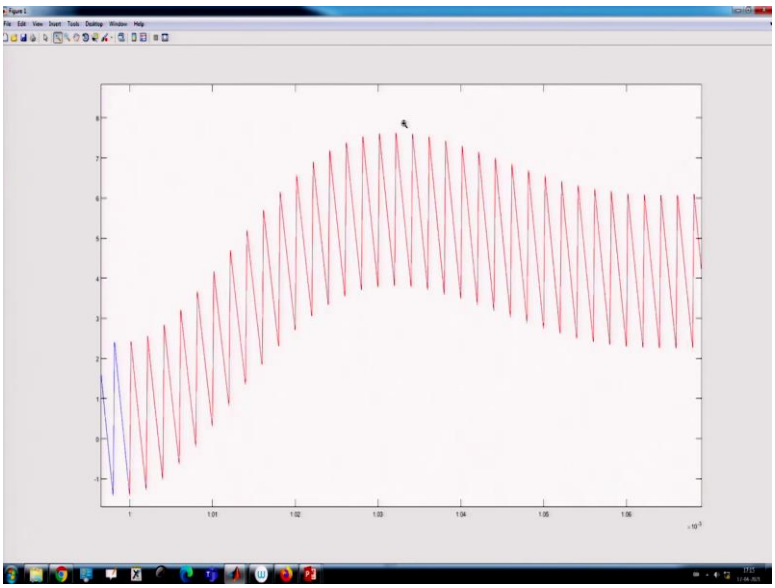


(Refer Slide Time: 41:12)

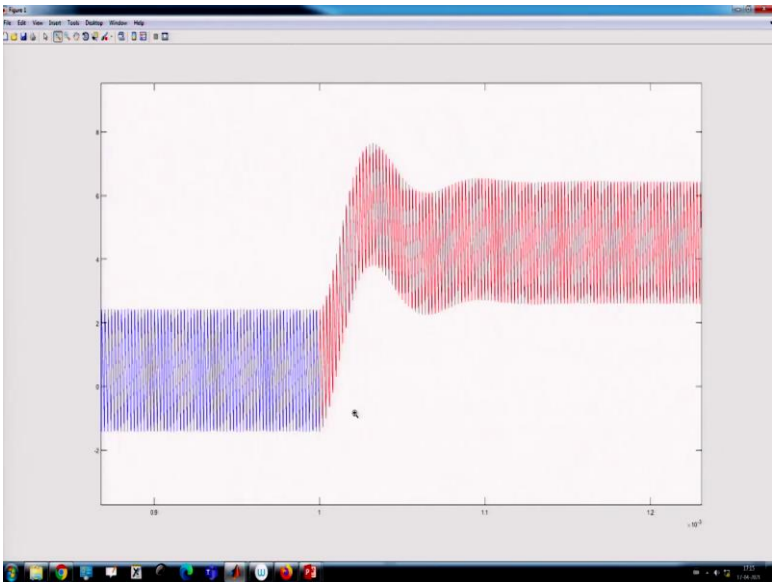


Now, it shows that simulation time the current also has changed and two different colour we can say that the transient actually scenario.

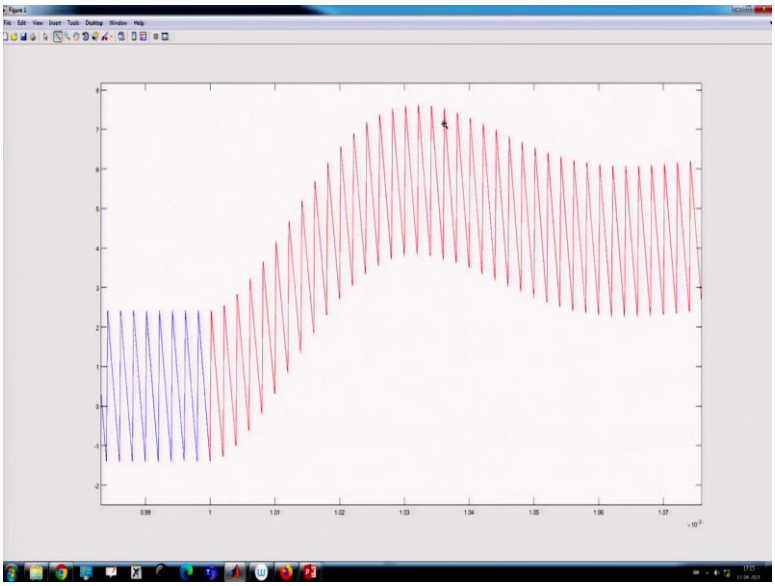
(Refer Slide Time: 41:14)



(Refer Slide Time: 41:19)

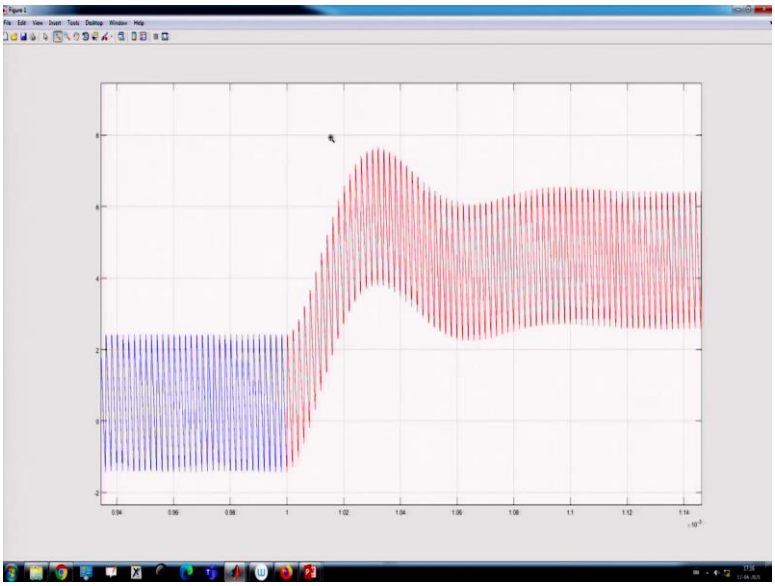


(Refer Slide Time: 41:21)



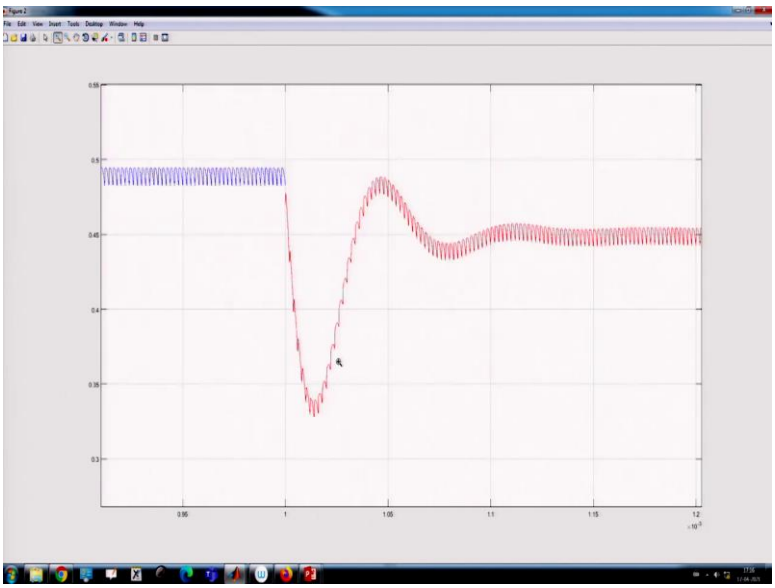
And by that way, we can simulate the load transient scenario, different transient scenario we can simulate here.

(Refer Slide Time: 41:31)

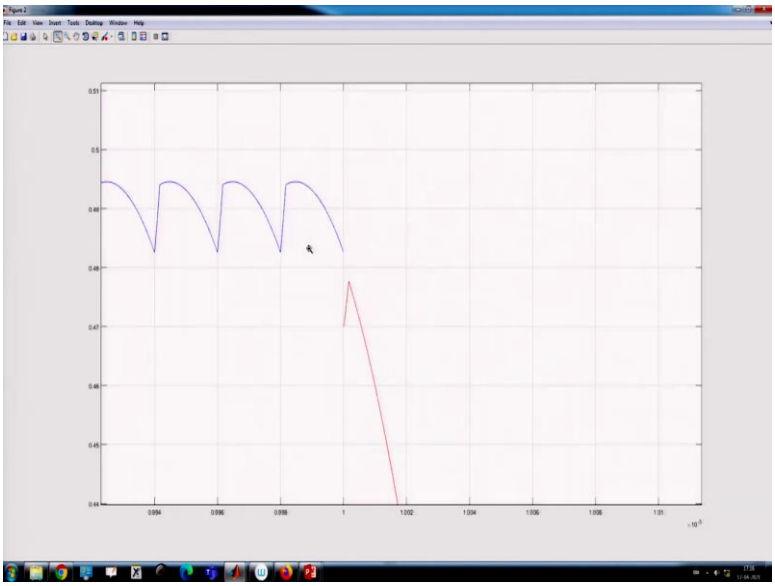


Load has undergo; undergone some load transient because we have change the resistance value.

(Refer Slide Time: 41:44)

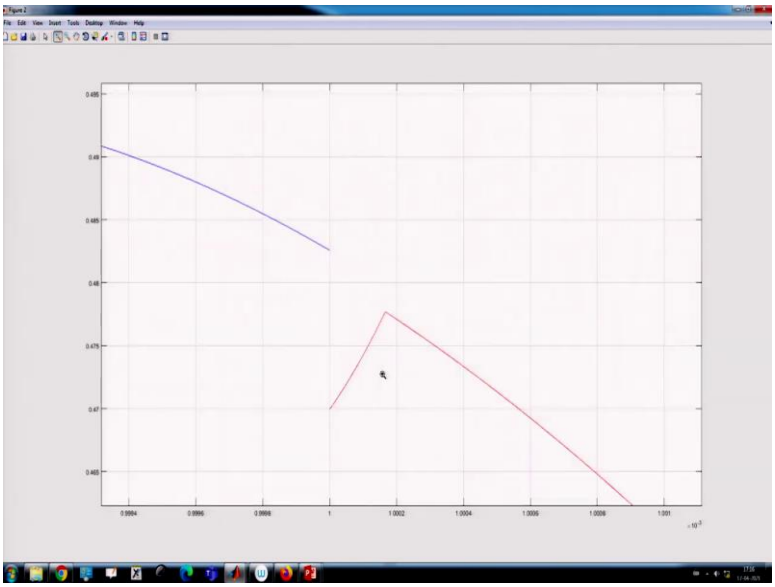


(Refer Slide Time: 41:46)



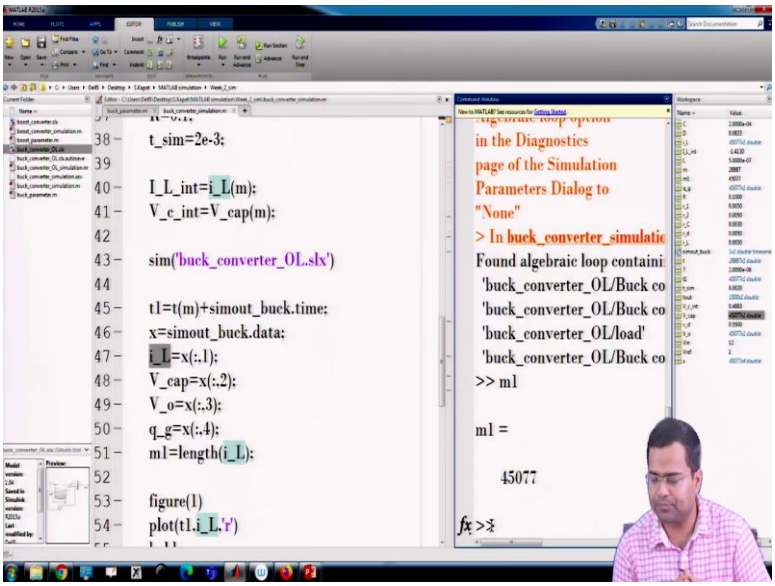
And output voltage, the jump in the output voltage whatever we can find from here, the output voltage there is a jump, this is due to the ESR, the ESR drop, there is a ESR drop ok. So, this ESR drop is.

(Refer Slide Time: 41:56)



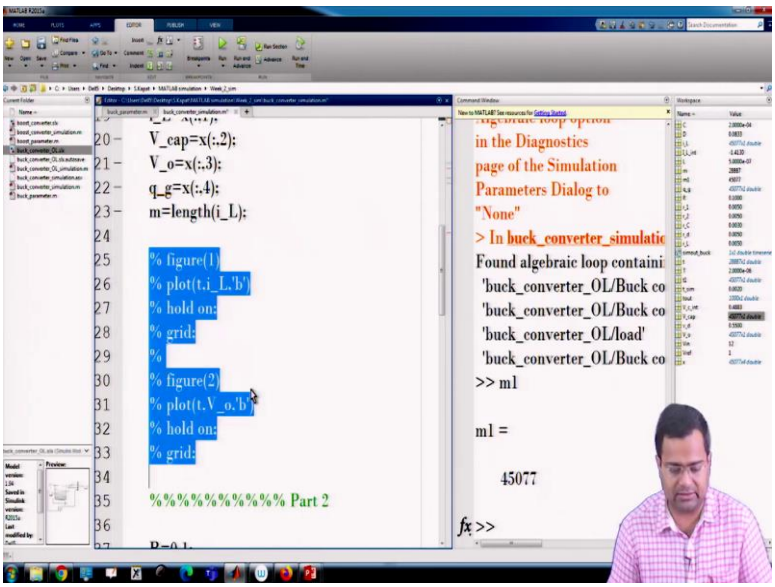
But now, how to because we are separately running, we have to combine together.

(Refer Slide Time: 42:16)



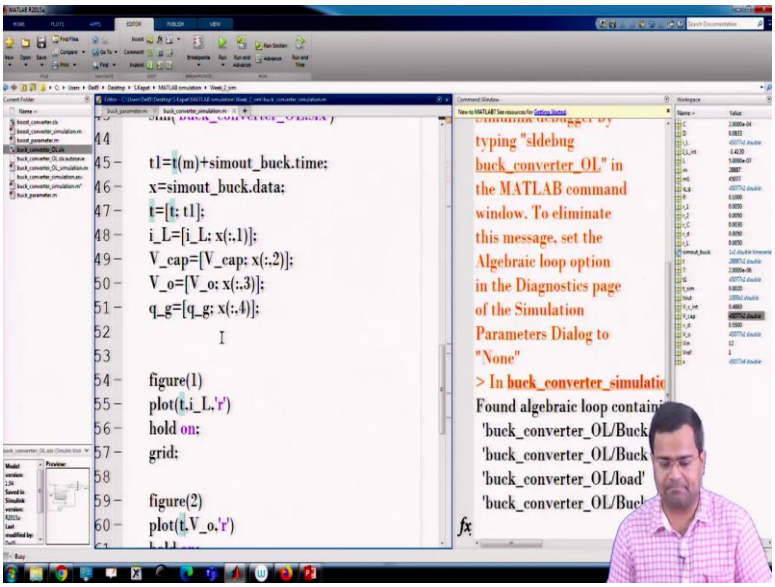
So, we can combine this x now, inductor current if we want to concatenate that means, we want to so, what is the t1 length? T1 like a m1 45000, this is my t1 yes. So, I want to plot, I can plot together, I do not I want to plot all these together, how can I do that? So, I do not want to plot here now; I want to plot together using a single so, I just comment on this.

(Refer Slide Time: 42:46)



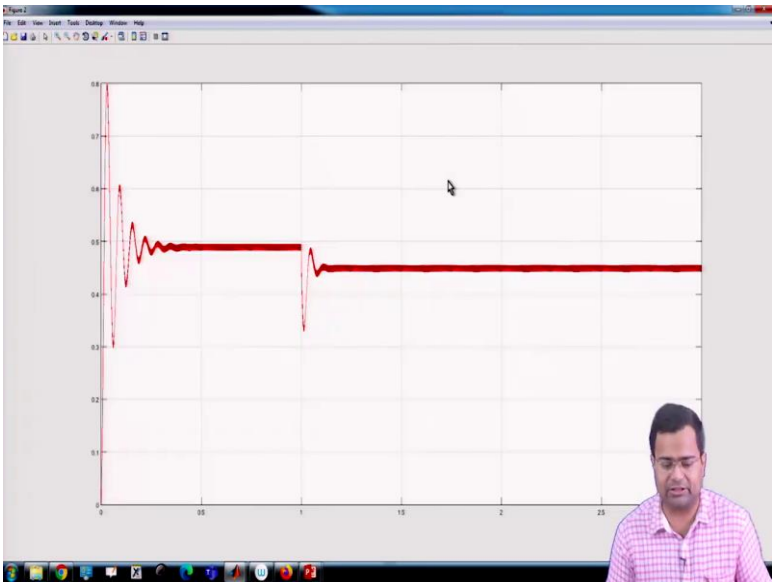
So, if I want that means we want to create a structure that matrix vector earlier inductor current where it finishes, then from there another column we will create so it will add together.

(Refer Slide Time: 43:06)



How to do that? It is simple ok. Here, I have to create this t this. Then t1 ok, this is my new t i L will be previous i L, then i this is called concatenation I can do it, concatenate, then here I can use V cap that means, previous V cap plus this V cap will get concatenated, then this V0 will get concatenated and this one q g, q g concatenated with this. After that we do not need. Now, we can plot the complete t that means, now you can plot t, t and let us same.

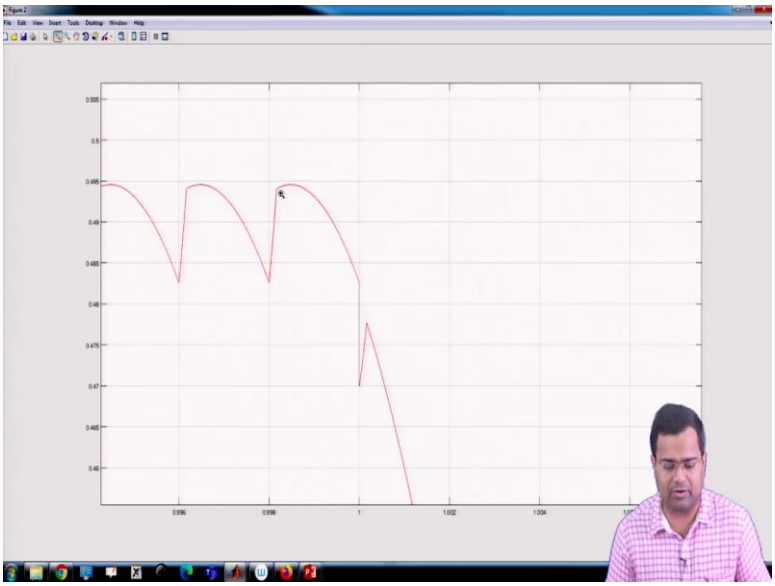
(Refer Slide Time: 44:22)



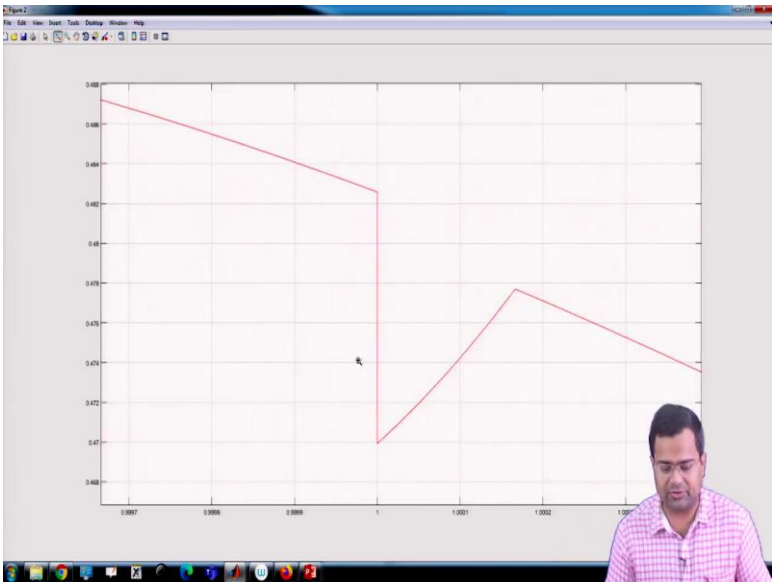
Now, it shows the complete simulation.

(Refer Slide Time: 44:25)

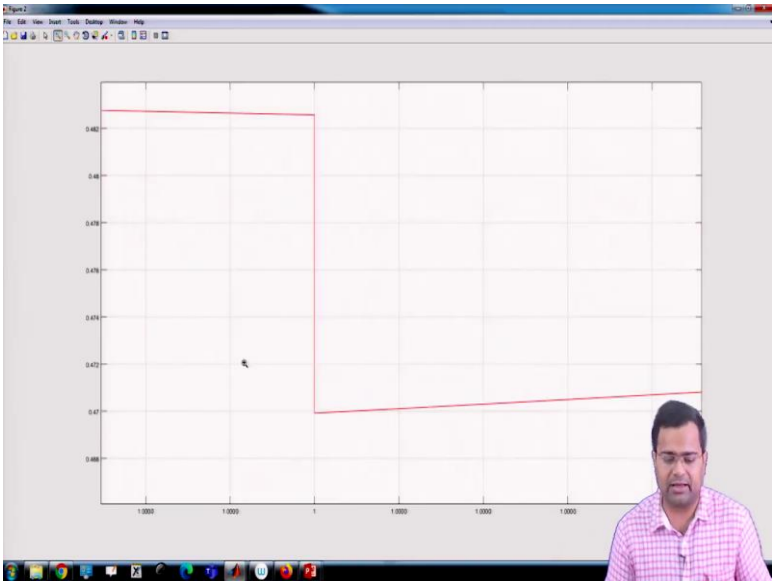
(Refer Slide Time: 44:27)



(Refer Slide Time: 44:29)

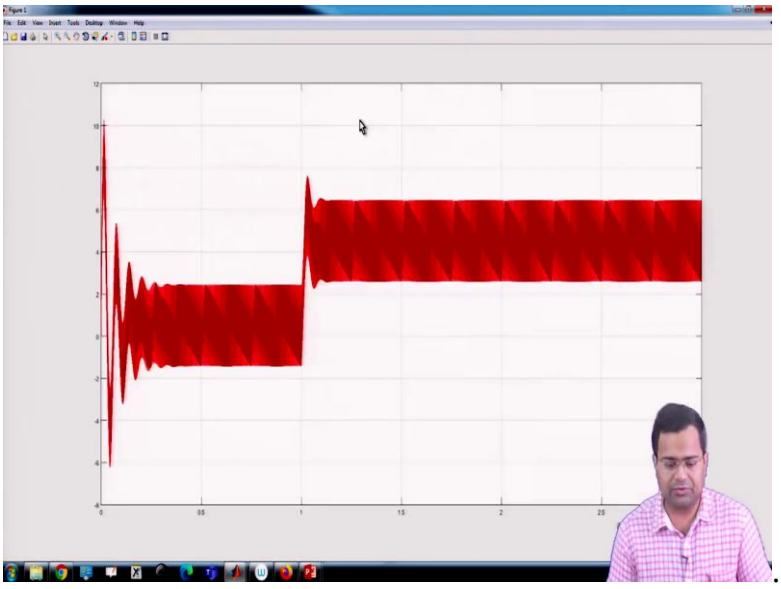


(Refer Slide Time: 44:31)

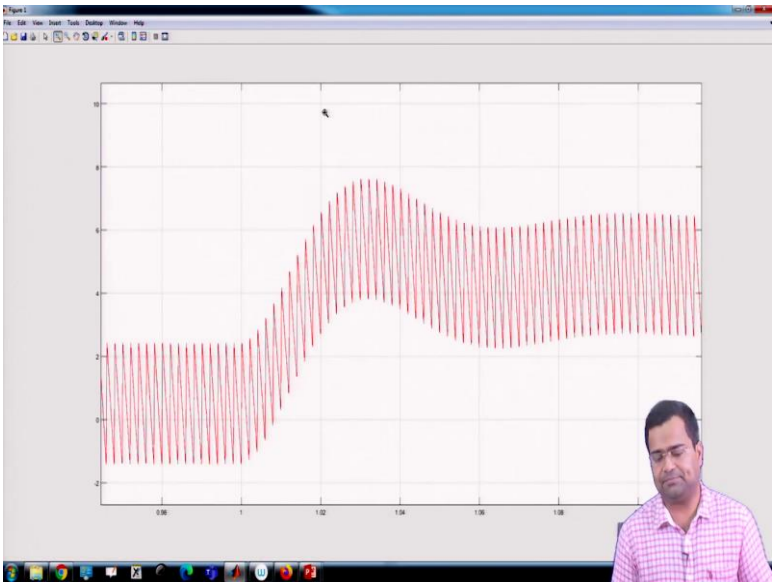


And it also join that discontinuous line where we got some discontinuity so, this was our point of discontinuity because we have clubbed together both the simulation and you know they can run.

(Refer Slide Time: 44:41)

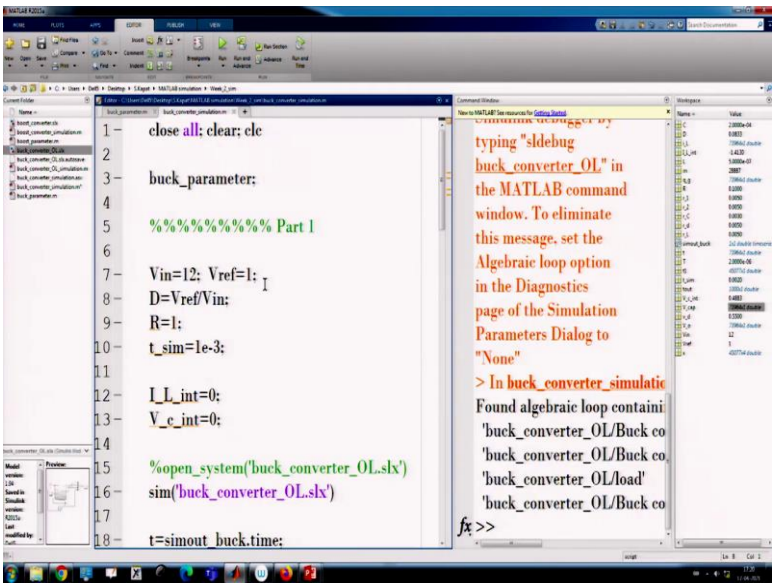


(Refer Slide Time: 44:46)



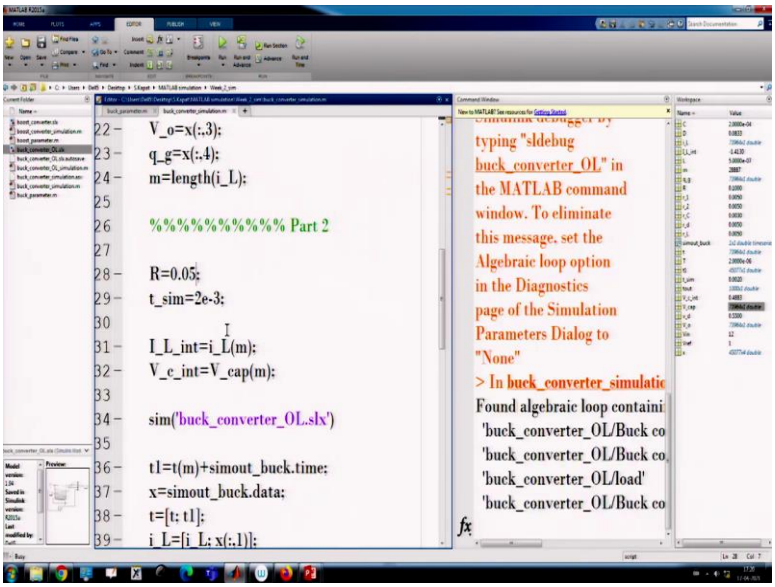
So, we have created a case study where the inductor current also we have merge them together that means, we learn that how to simulate a transient case study, this is in open loop, but we can do it in closed loop, there is no problem. Now, we will go to the Simulink block and changes. So, that means, we can create transient case study and we can make simulation.

(Refer Slide Time: 45:26)



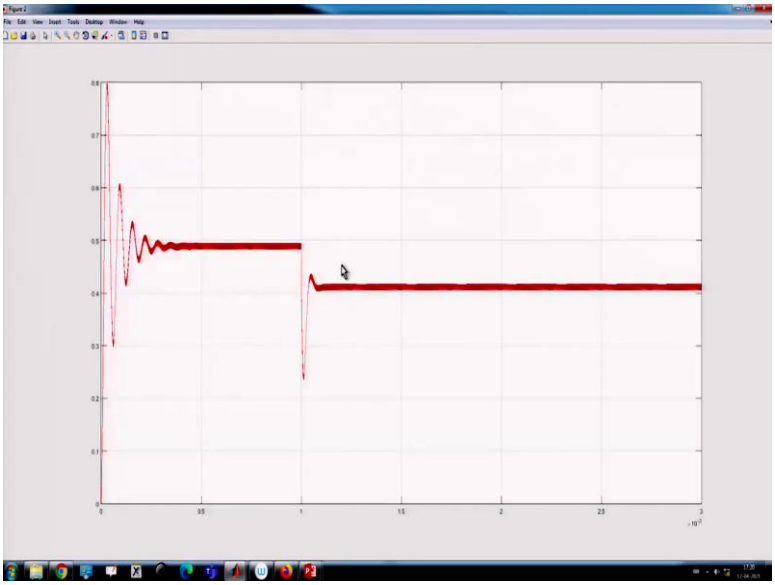
So, we can now remove all these plot so, part 2. So, we do not want to plot part 1, part 2 separately, we want to plot it together. Now, we can play with different simulation time you know when do you want so, all these we can do it.

(Refer Slide Time: 45:33)

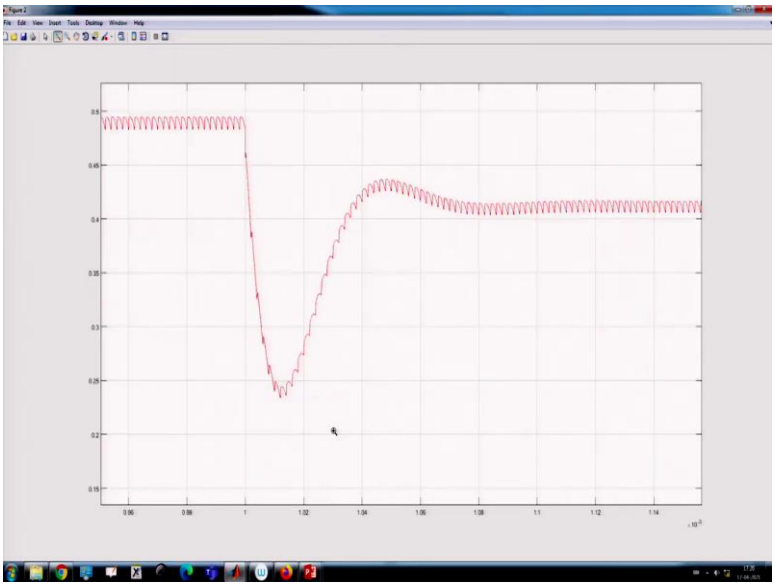


Now, we have to do closed loop control. So, load resistance we have change, we can change even further down, we can run it so, it will show.

(Refer Slide Time: 45:36)

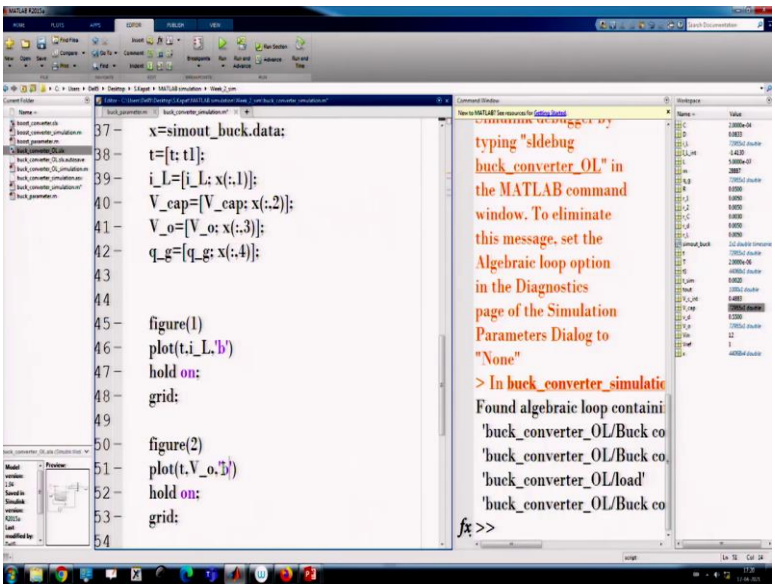


(Refer Slide Time: 45:42)



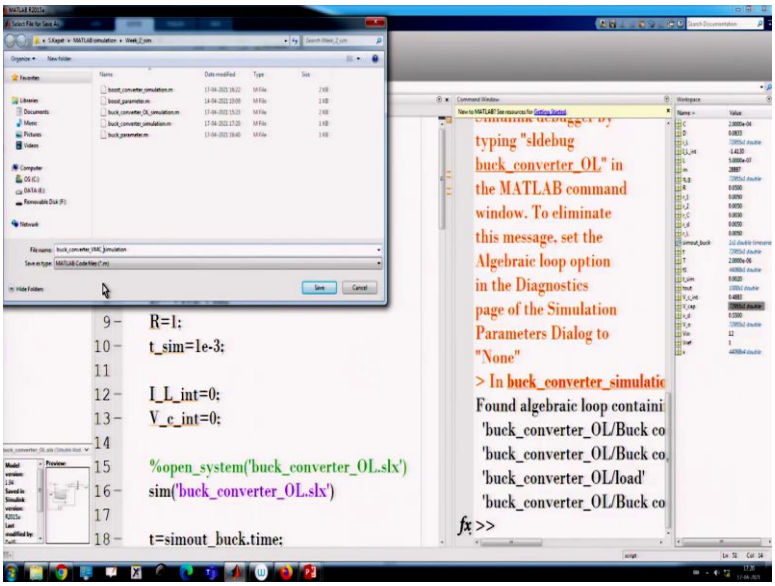
And you say that the simulation time is much faster and if you run the same thing in Simulink, it might take more time because you can save lot of simulation time here ok.

(Refer Slide Time: 45:53)



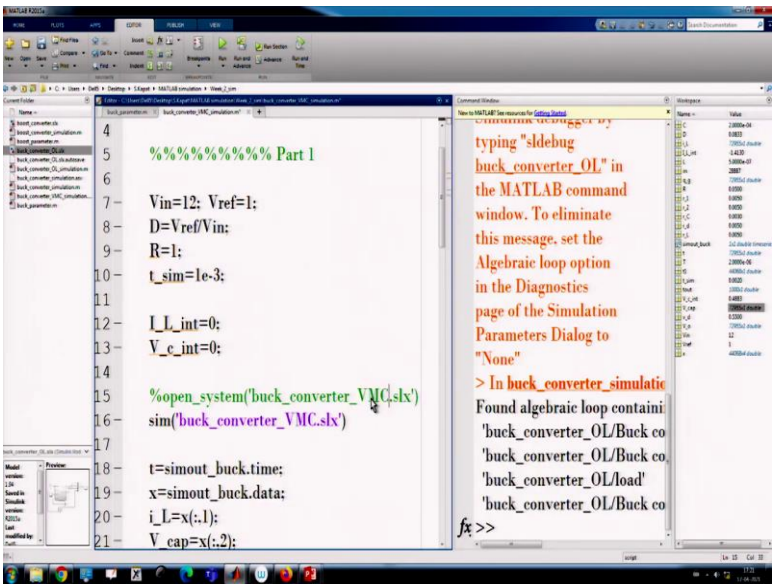
So, now this I am showing.

(Refer Slide Time: 46:04)



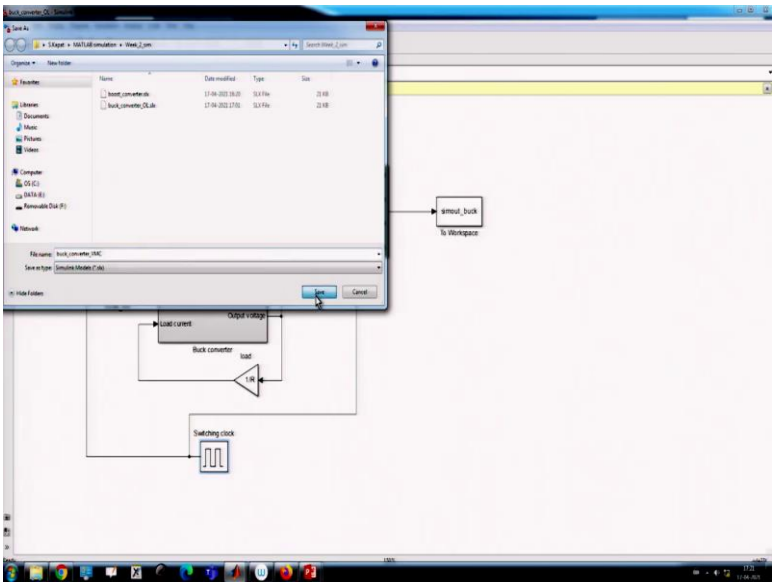
Now, the same file I am going for a closed loop so, I will call it as a save as you know buck converter simulation like a voltage mode control ok.

(Refer Slide Time: 46:20)



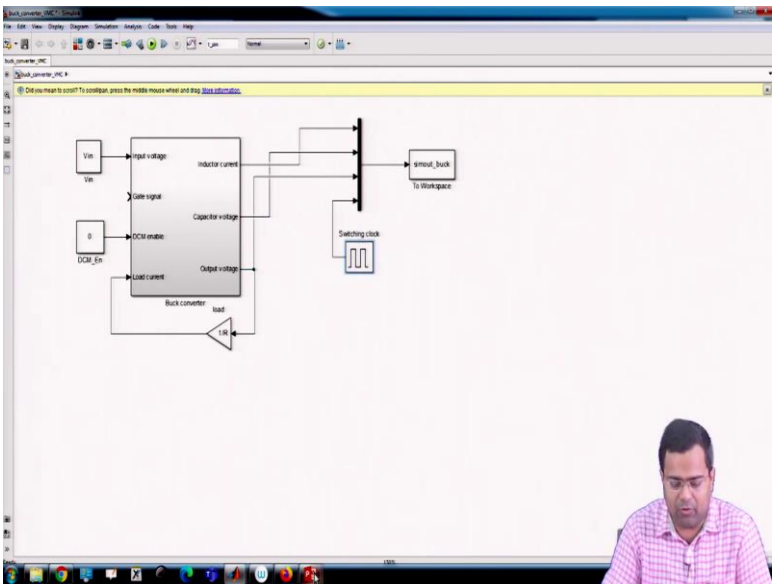
So, in this voltage mode control simulation, I will use another file name VMC, but I have to make changes there VMC.

(Refer Slide Time: 46:35)



So, because we are using another excel file so, we have to go and save as here it should be VMC ok.

(Refer Slide Time: 47:05)

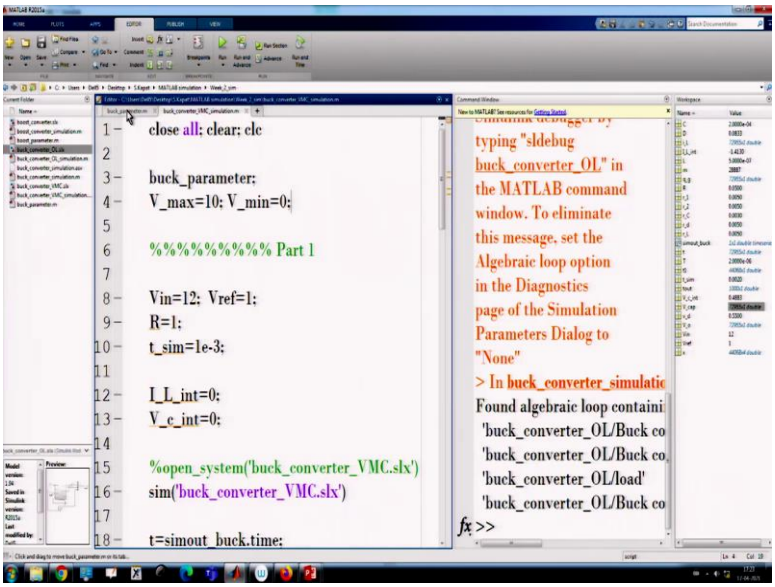


Now, we have created a simulation environment for voltage mode control. You need to see the closed loop control ok. So, far we have to check with the open loop now, we have to go for a closed loop. So, in a closed loop, now, we will not operate the gate signal using a fixed duty ratio. So, what are you going to do? Now, if you go back to our ppt so, we need to create something like this that means trailing edge modulation and in this

modulation; we want to use the control voltage and we want to compare a sawtooth waveform.

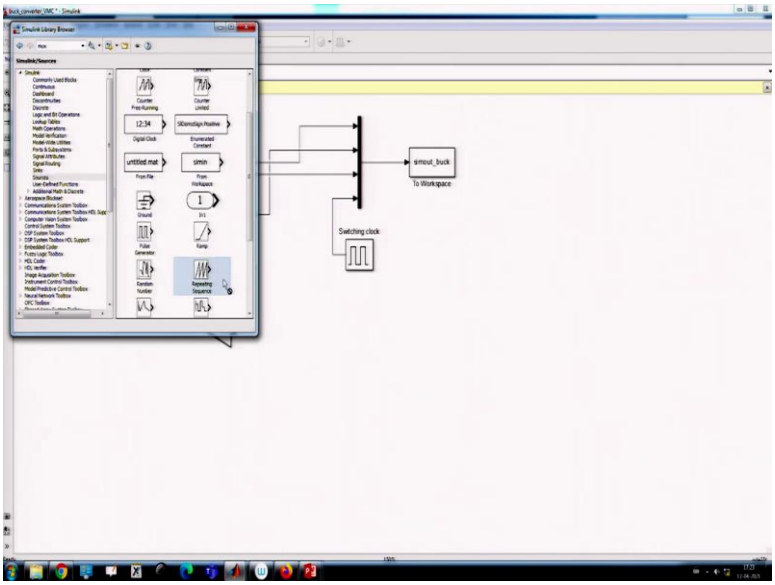
So, this sawtooth waveform, we need to fix some upper and lower limit ok the sawtooth waveform. So, where we call it as a V max ok and we can start here, we can call it as a V min and this is my time period T, I can set it ok V max, V min, T. Now, how to go? We want to go to simulation, yes.

(Refer Slide Time: 48:10)



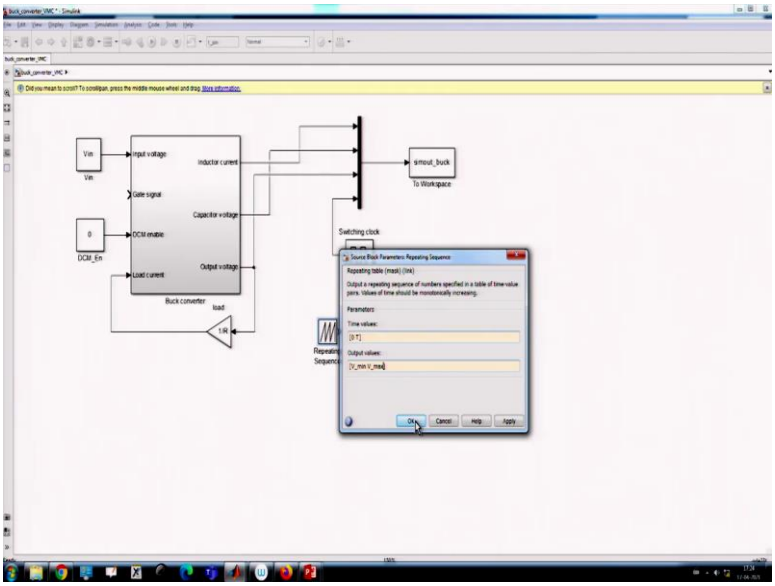
So, before that now, we need to define. So, the input voltage is given. I do not want to change because you can use this Vref, Vin whatever I want. I did not want to use a separate duty ratio because now, I am going for closed loop control. What we have to do first? This means, we have to set some other parameter which is my V max; V max equal to let us say 10 volt, V min is equal to 0, this is for m and time period is already actually defined. So, time period is defined like a 2 microsecond. So, this, this, this is enough.

(Refer Slide Time: 48:53)



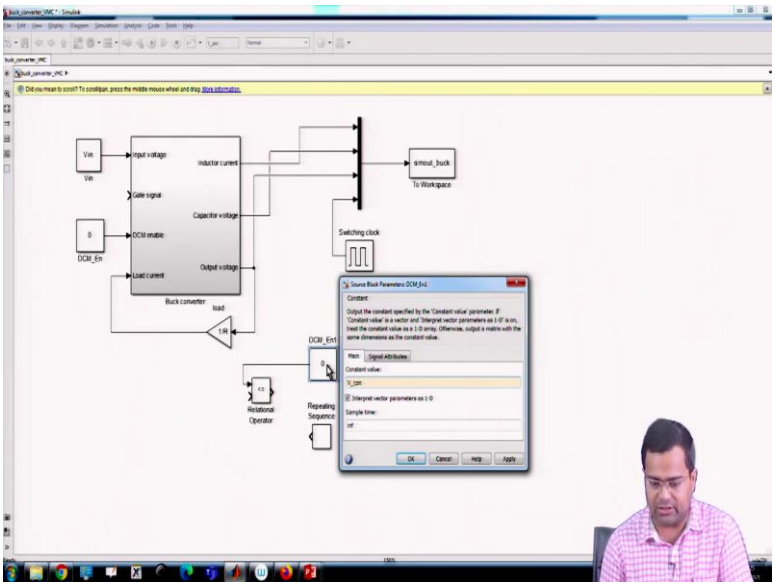
Now, what next we have to do? Go to the library, sources, there we will find this sequence, periodic sequence.

(Refer Slide Time: 48:59)



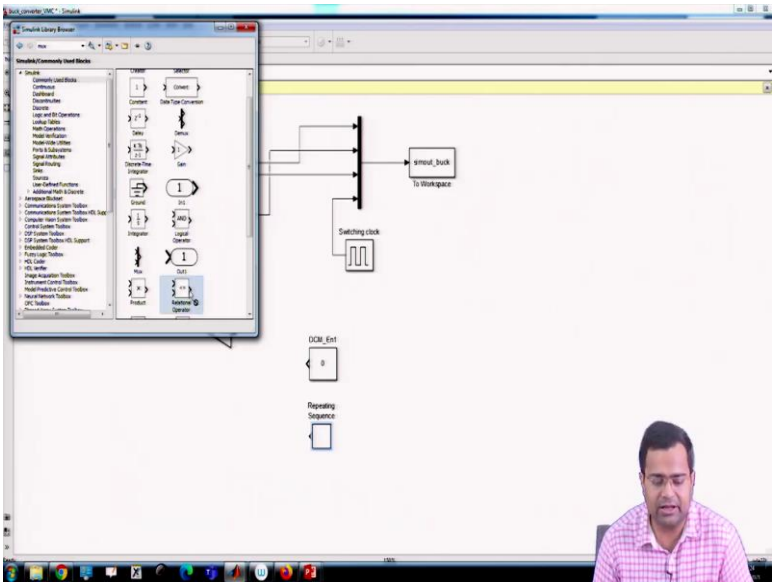
And if you go, the time will be T, it will from 0 to T it will repeat and it will change from V min to V max ok, but we need we are yet to set.

(Refer Slide Time: 49:17)



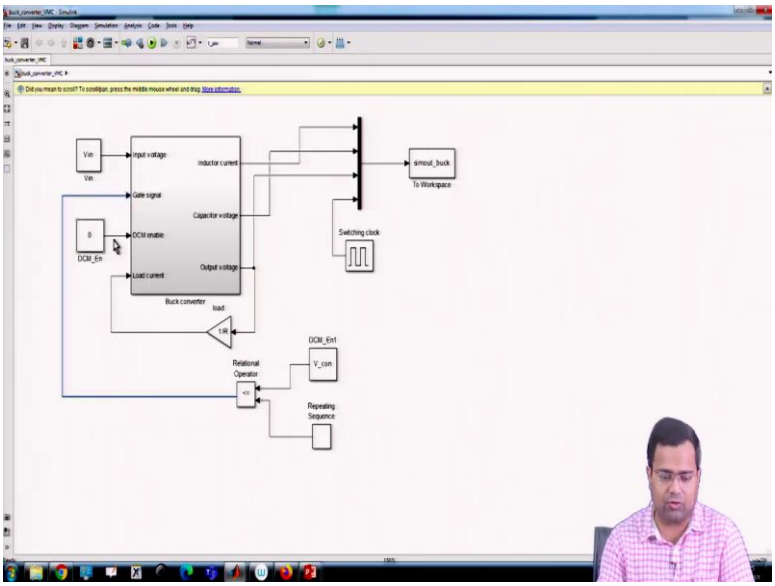
What is next? That means initially, before we go to voltage mode control, we want to provide a control voltage that means, the constant quantity ok constant quantity.

(Refer Slide Time: 49:42)



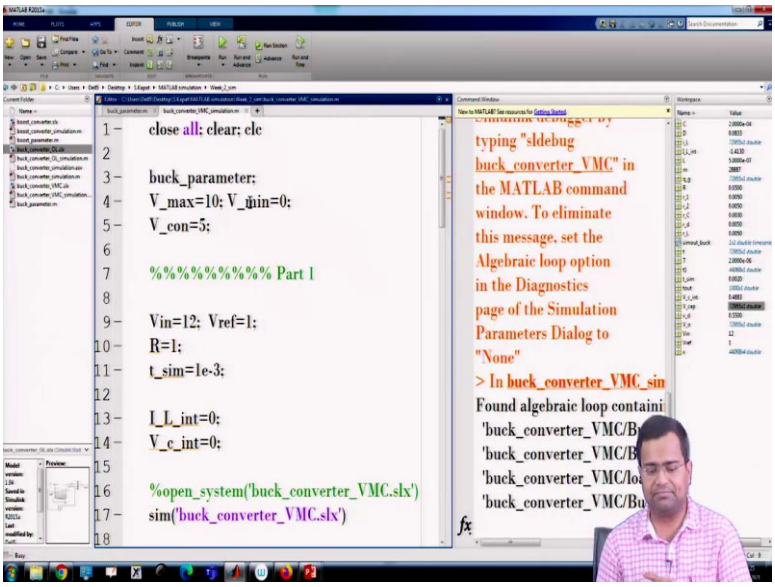
And we want to use a comparator that means, commonly use block, there will be something like a relational operator. So, if the first output this one is smaller than the second, it will go high that means, if my constant which let us say I am setting like 5 volt, let us say I am using V con, I could define what is V con.

(Refer Slide Time: 50:10)



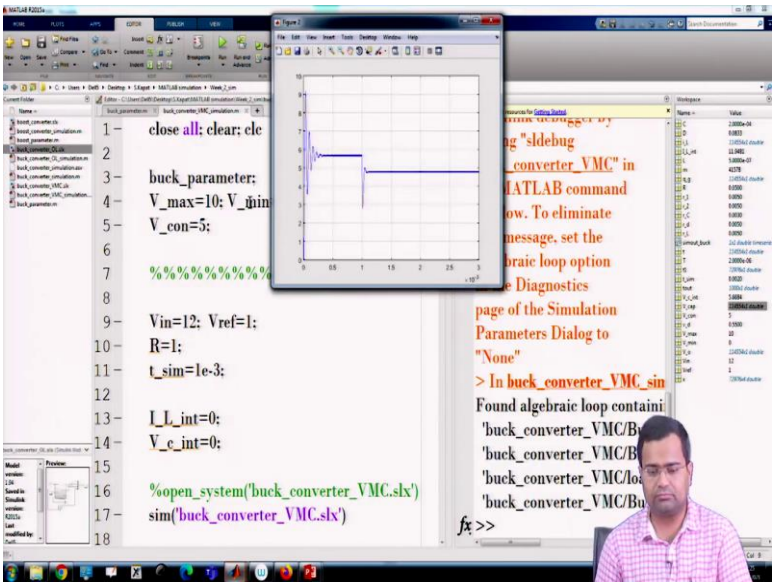
V con is smaller than this ok, then; smaller than this, then I will generate a gate signal.

(Refer Slide Time: 50:38)

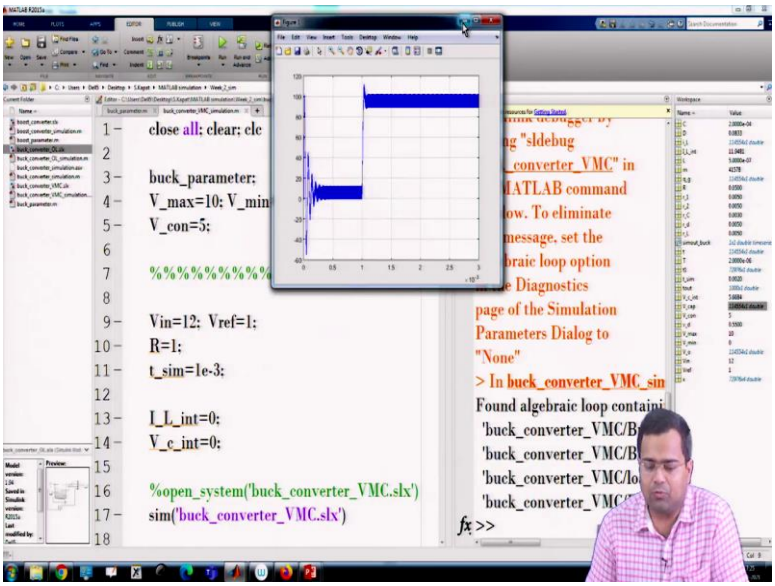


So, we are not; we are now; we have not yet used the closed loop, but just we are generating where we need to define what is my V con. V con equal to 5 volt. Now, let us check what happen? Yes.

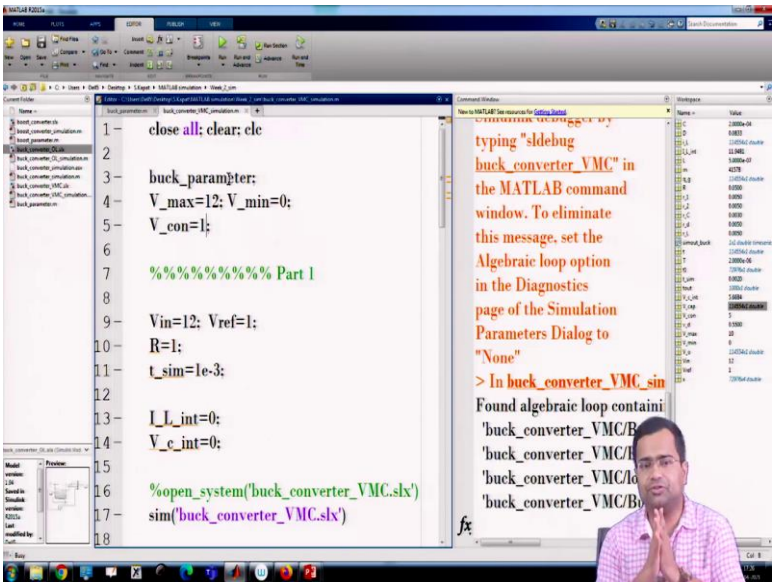
(Refer Slide Time: 50:47)



(Refer Slide Time: 50:51)

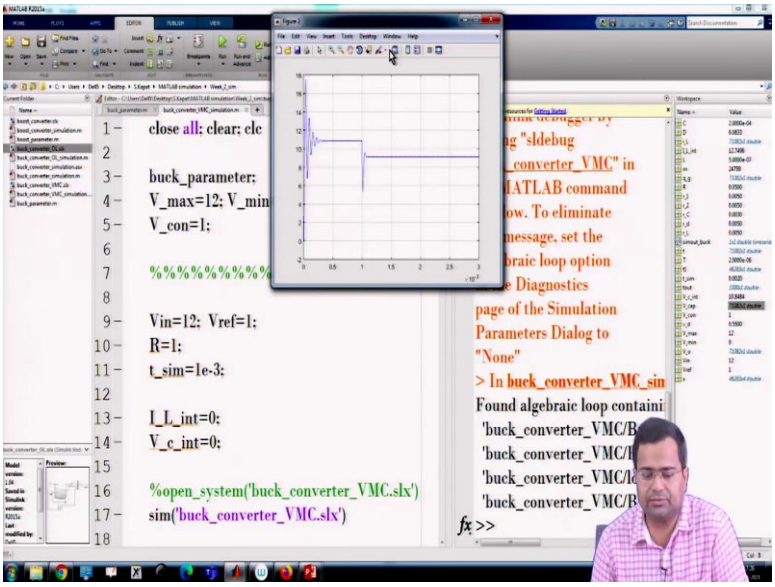


(Refer Slide Time: 51:07)



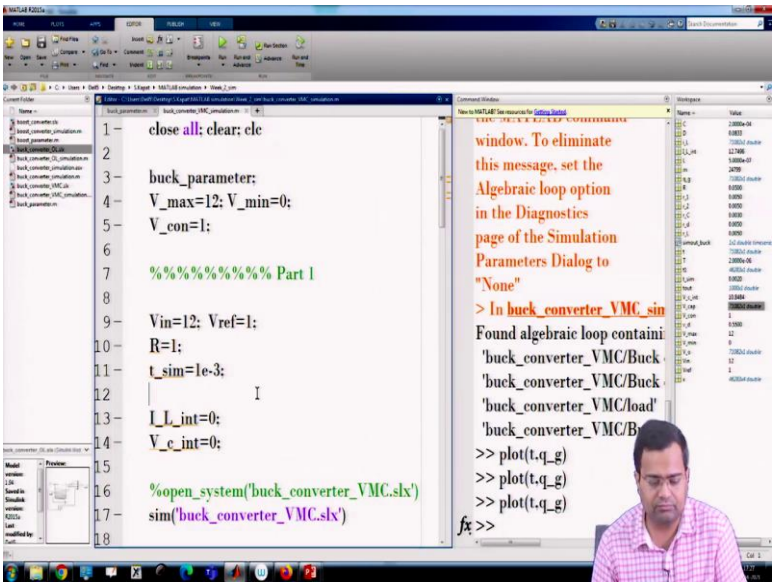
Now, the output voltage has gone high, it is too high because you are using 50 percent duty ratio, but V con what should be the V con? The control voltage you know in order to achieve the duty ratio so, let us say if we set 12 volt here, 12 volt and 1 volt so, you can set V con to be 1 volt, then duty ratio will be accordingly generated.

(Refer Slide Time: 51:12)



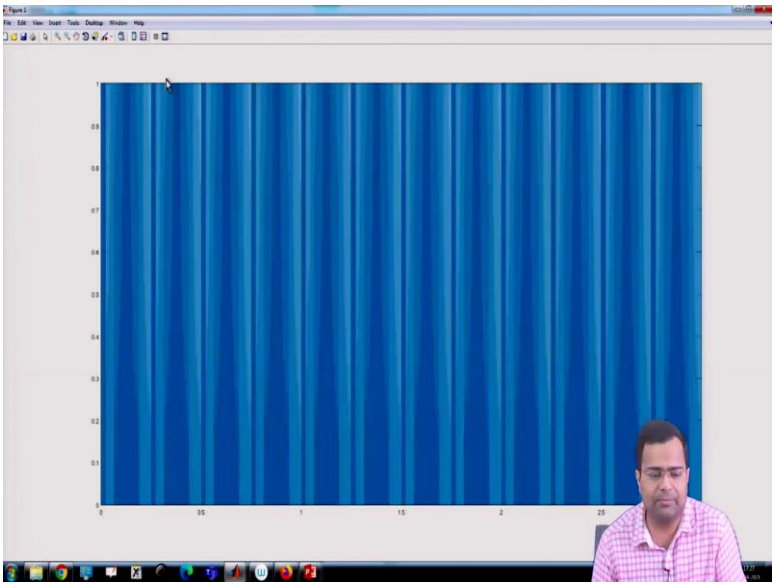
So, 12 to 1 volt that means, my sawtooth waveform is going to 12, V con is if you want to check the gate signal ok.

(Refer Slide Time: 51:40)



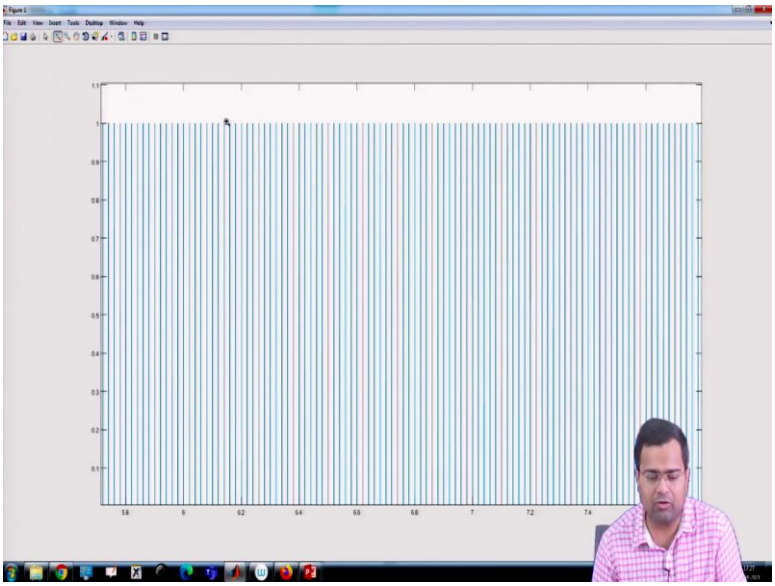
Let us plot t comma q g k yeah.

(Refer Slide Time: 52:05)

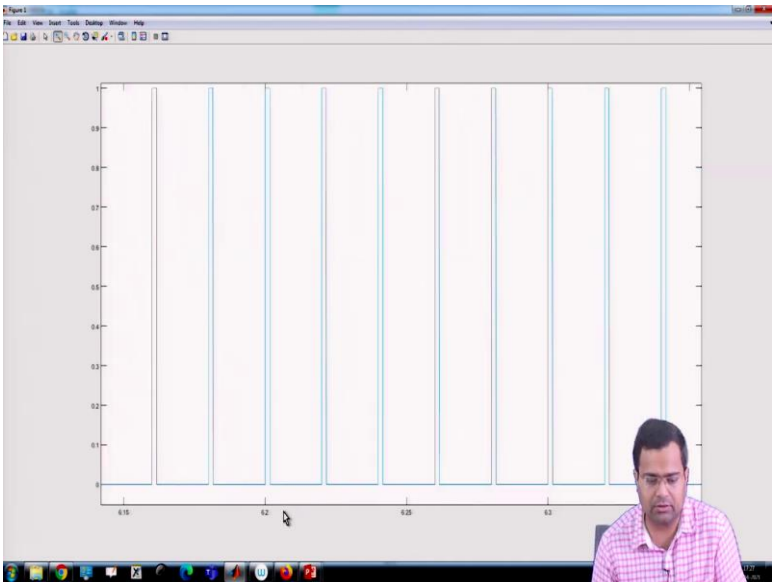


So, if we take the gate signal, I want to see what happen actually?

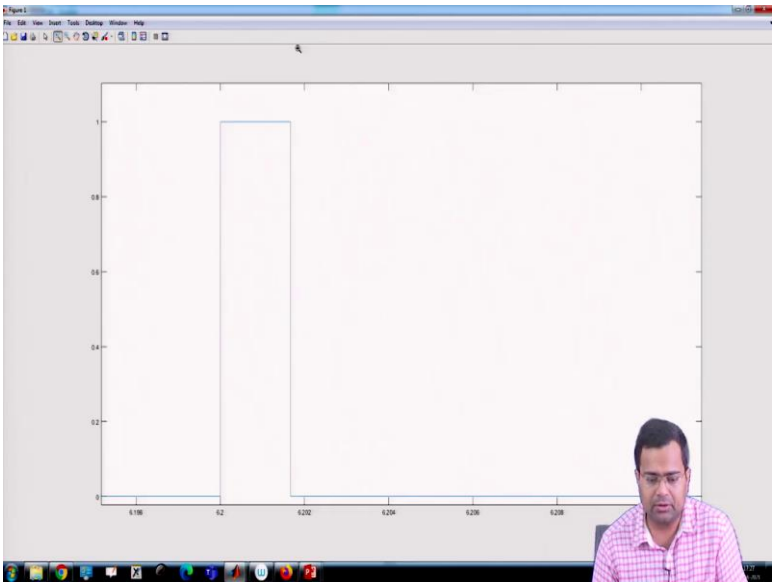
(Refer Slide Time: 52:12)



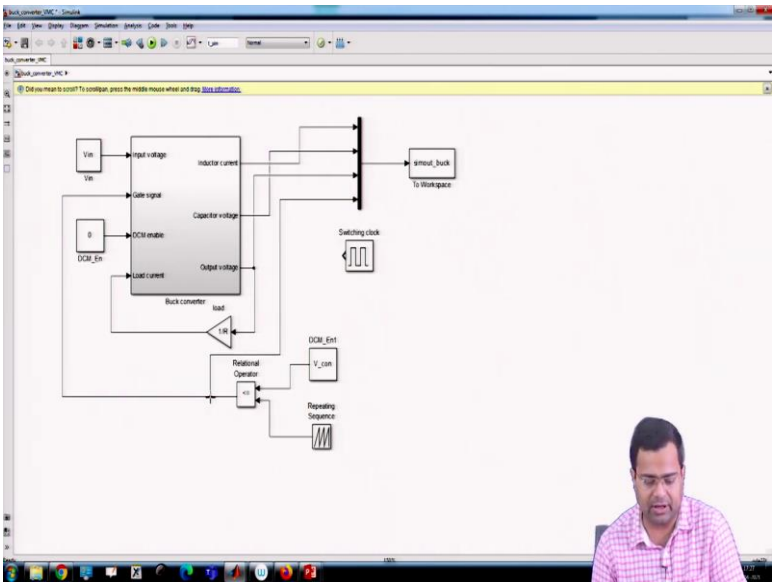
(Refer Slide Time: 52:14)



(Refer Slide Time: 52:18)

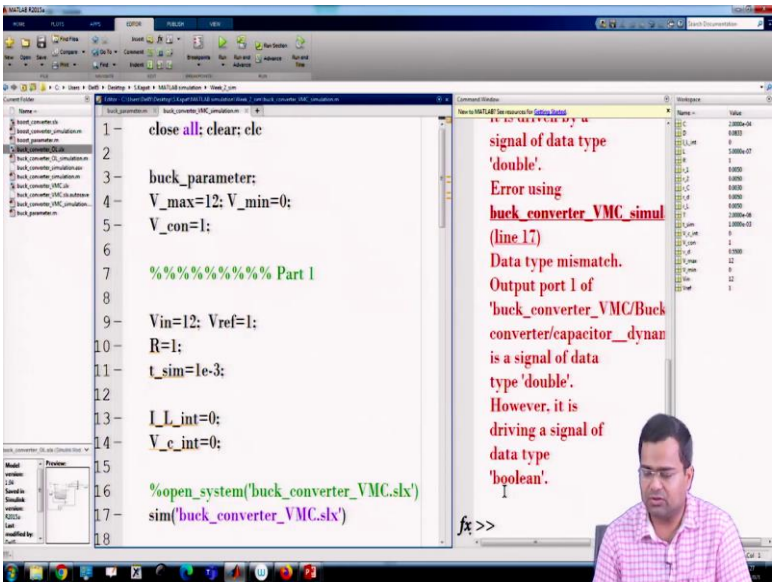


(Refer Slide Time: 52:25)



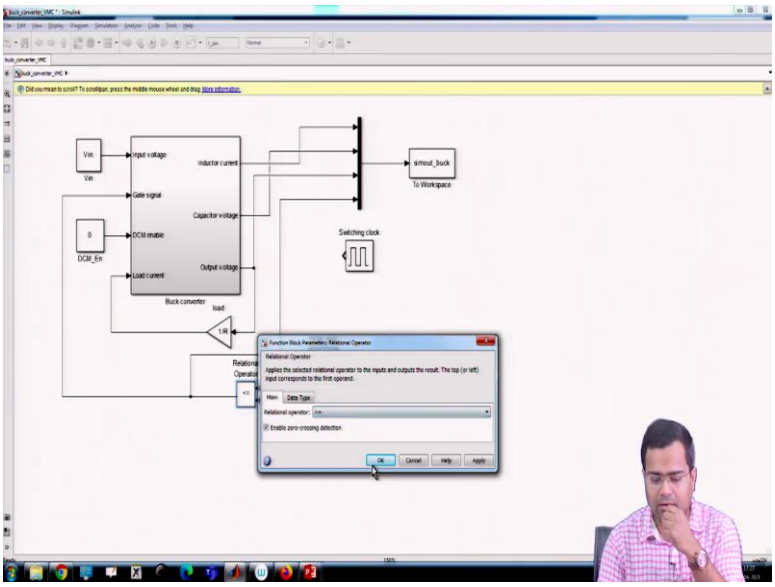
Maybe the logic is yeah, it is a very low duty ratio, but then why low duty ratio the q g from where you are tapping the q g? We should tap from here ok, let us run it.

(Refer Slide Time: 52:34)



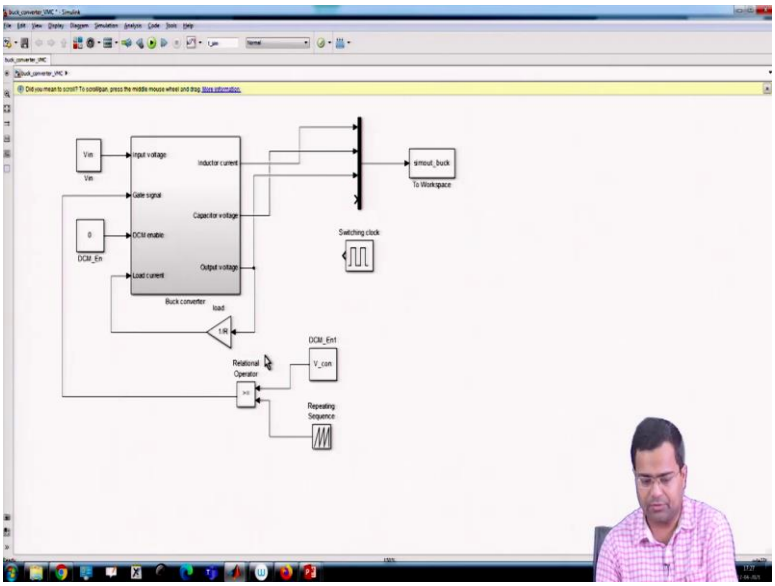
And so, driving signal ok.

(Refer Slide Time: 52:50)



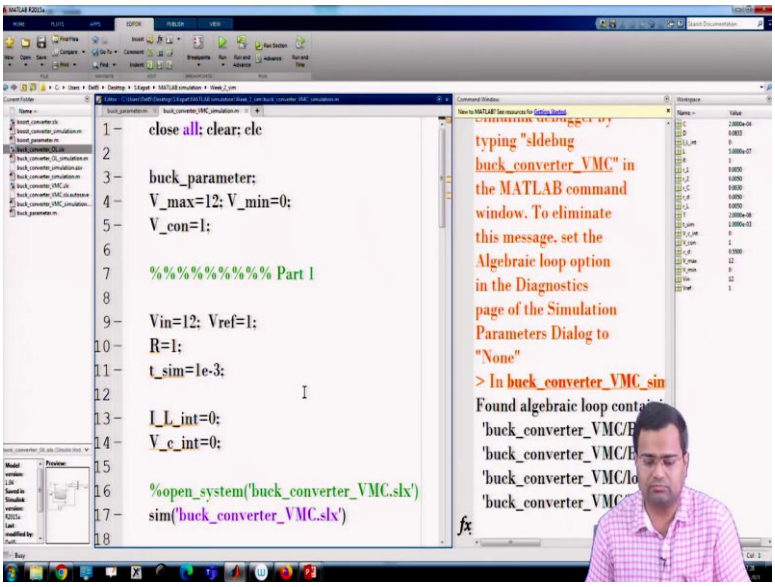
Maybe we need to put a score a driving signal. So, q g gate signal ok relational operator.

(Refer Slide Time: 52:56)

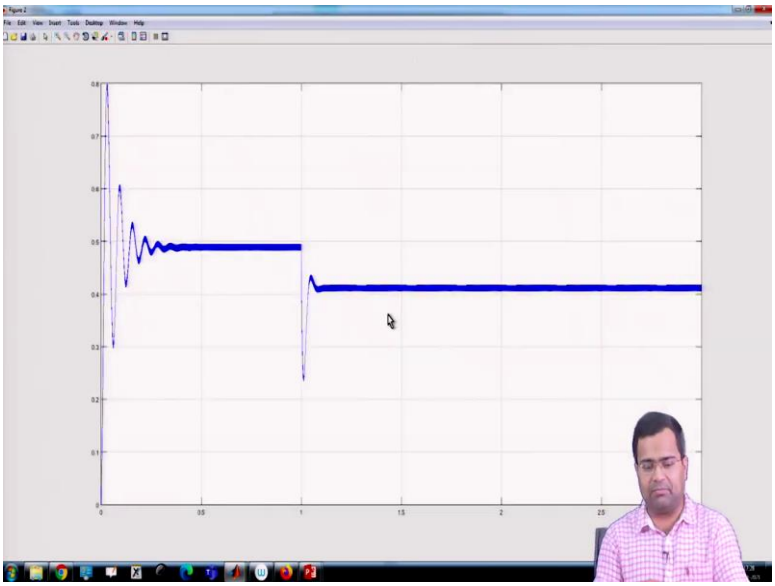


Should it be ok, let me check whether there is any logical change or not.

(Refer Slide Time: 53:07)



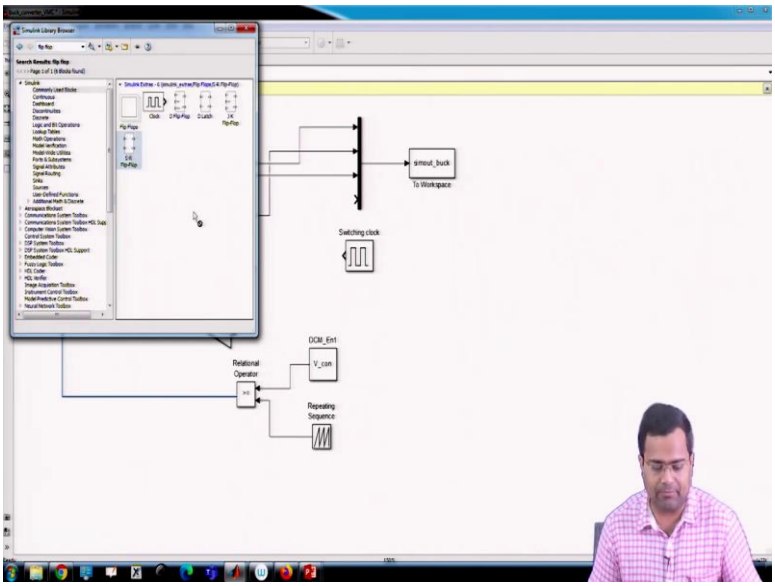
(Refer Slide Time: 53:09)



Yeah, I think there was a change logical problem because what was the initial logic? If V con is it should be high when the V con is higher, but I took the reverse logic, V con as long as V con is higher than sequence voltage because if we go back to this waveform, as long as this V con you know because if we set V con somewhat here so, it is higher, then it should give the on-pulse, this is the V con, but we have not yet incorporated the latch circuit here ok. So, let me erase this part ok.

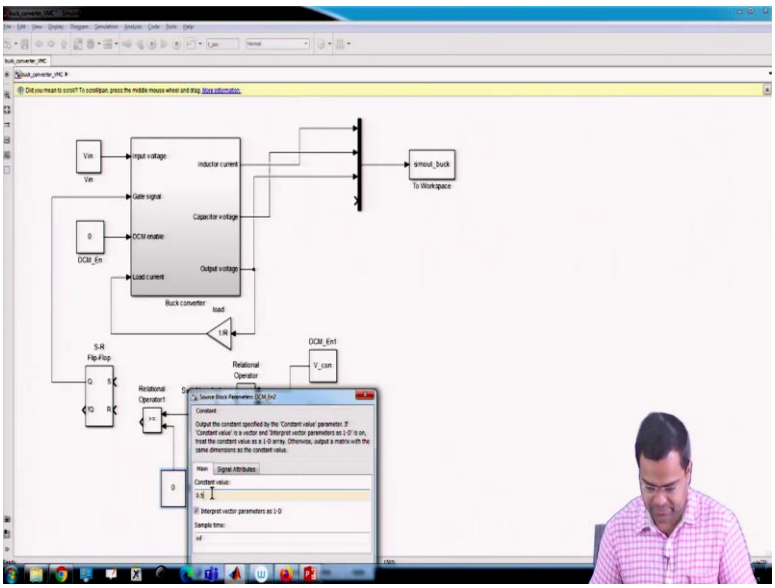
Now, going back to our logical block. So, we have set a V con logic, but now, this is not enough that means, it will be high, when it will reach certain value.

(Refer Slide Time: 54:11).



Now, we want to introduce our R-S latch that means, latch circuit. So, that flip-flop so, R-S flip-flop we want to introduce.

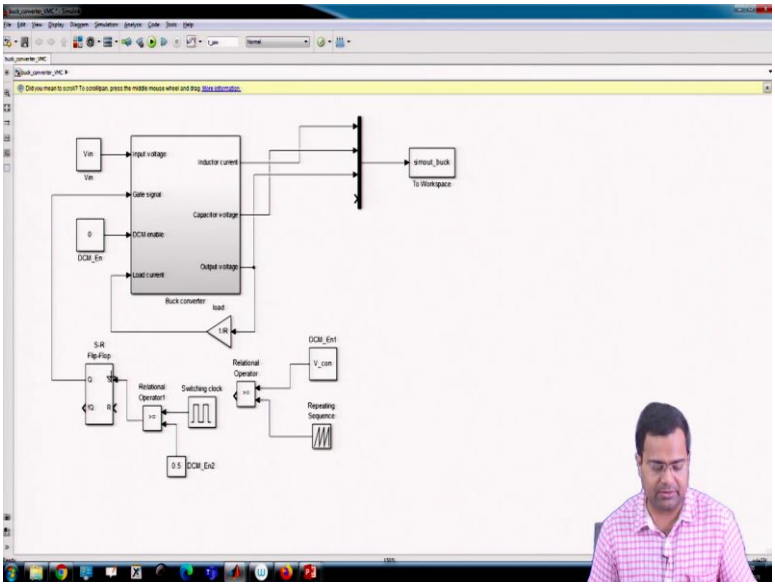
(Refer Slide Time: 54:19)



And this R-F flip flop because we want to implement our trailing edge modulation technique. What is the set pulse for this flip-flop? The set pulse for this flip-flop again, flip-flop cannot take, there can be double if you try to run it because this is the double data, data type is a double, but it require a Boolean function so, we need to set another

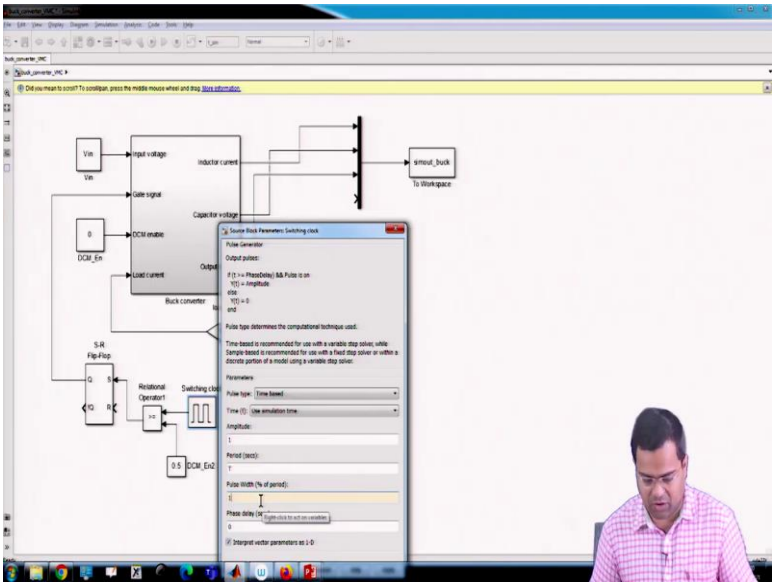
such you know relational operator that means, as long as this is higher than, greater than 0 or something like that so, greater than 0.

(Refer Slide Time: 55:25)

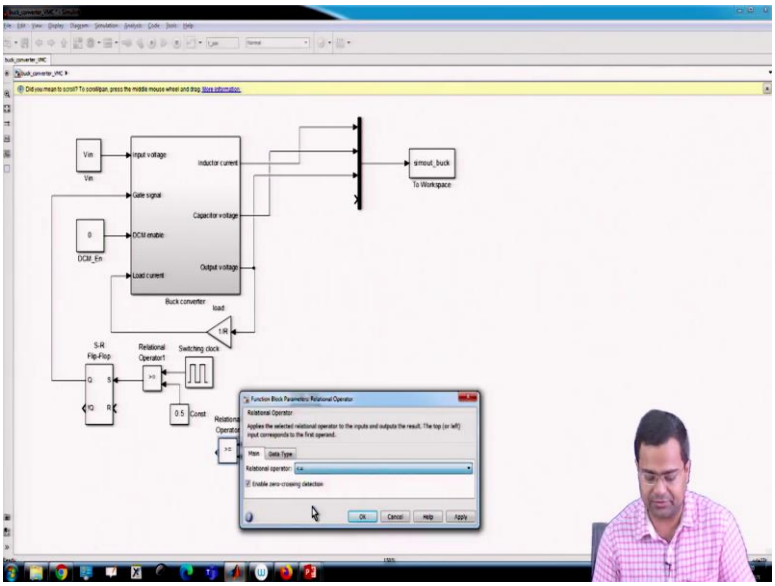


So, it is greater than 0, it will be high so, you can set like a 0.5 something like that yeah. So, this is my set pulse; set pulse.

(Refer Slide Time: 55:36)

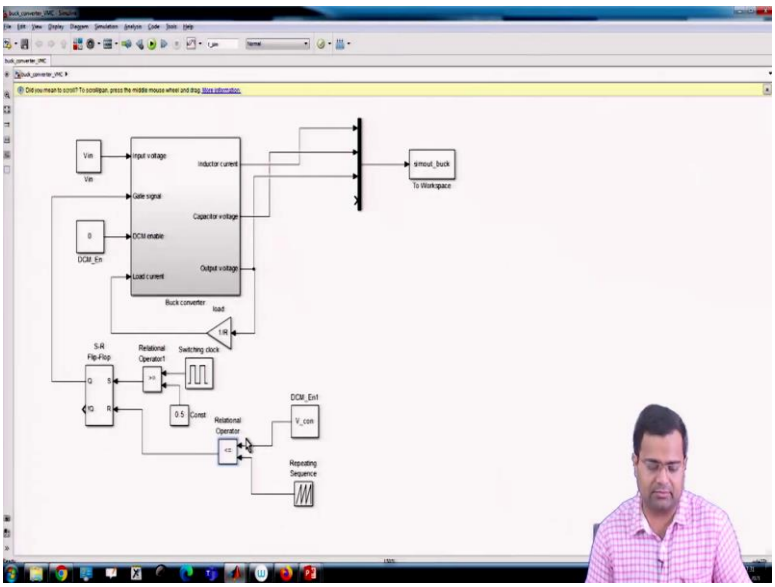


(Refer Slide Time: 55:38)

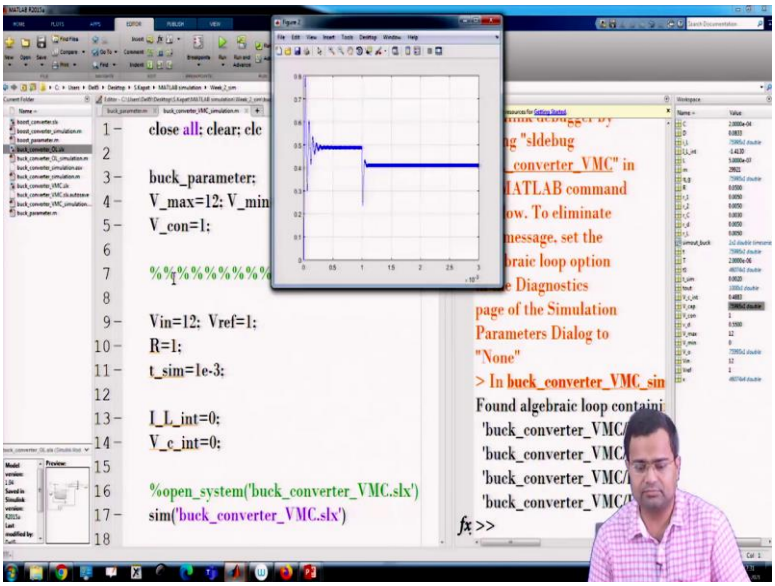


And here, I can use just 1 percent because it is just the edge, we require the clock; clock edge ok fine, this is my DCM, that is my constant that means, this is something like constant and now, I have to use a relational operator and here, it is used to reset that means, we have to reverse the logic. As long as V con is above, it should be high so, it should, we should now reverse the logic because in order to operate this.

(Refer Slide Time: 56:07)

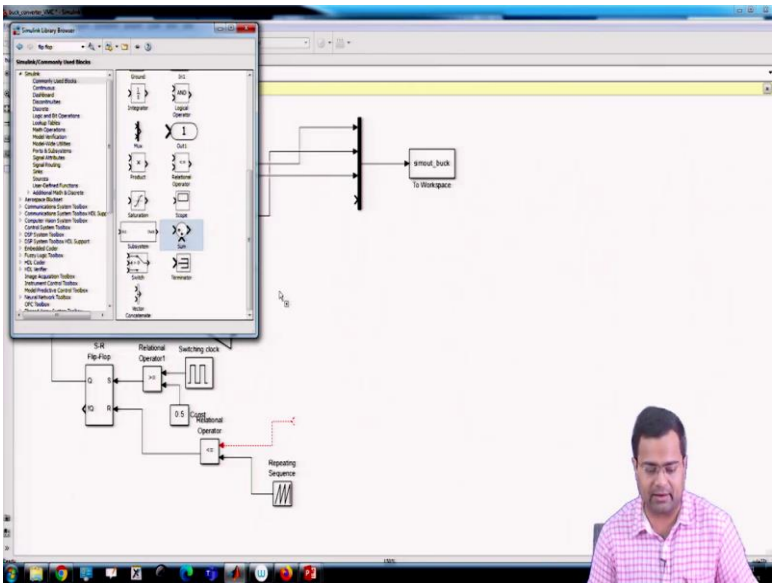


(Refer Slide Time: 56:16)



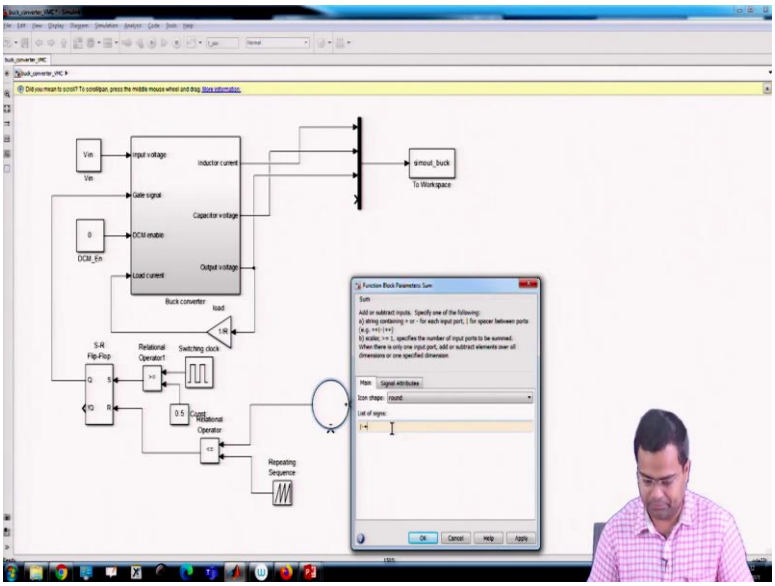
Now, we can start the simulation and check. Yes, same thing is happening we are just. So, this is like you know our controller where we have not yet use the closed loop control, we are just using.

(Refer Slide Time: 56:25)



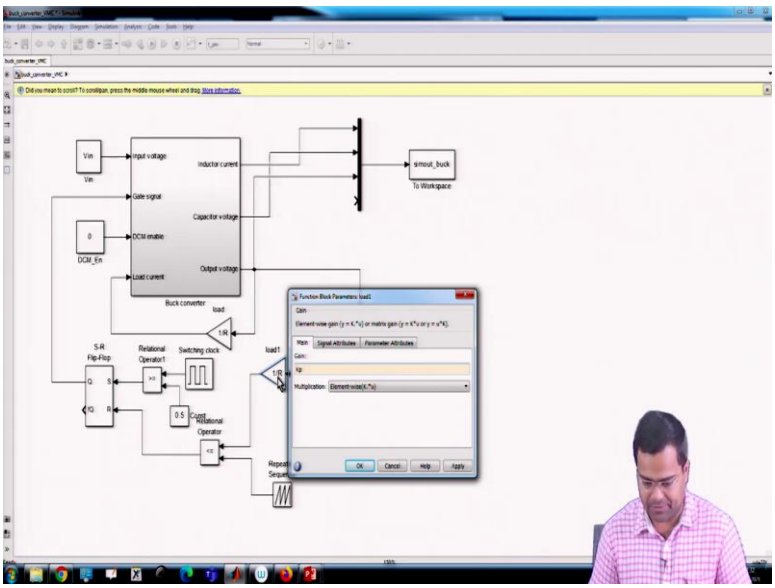
Now, this control can be modified by placing a error voltage that means, we need to consider one you know the traditional block feedback like feedback logic.

(Refer Slide Time: 56:42)



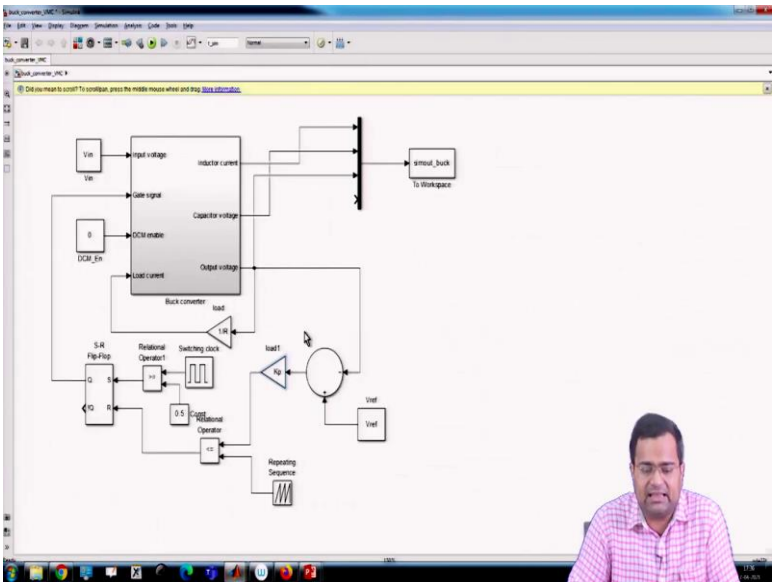
So, this is my feedback logic and this output will not go directly so, we need to so, this is 1 plus other is minus sorry ok.

(Refer Slide Time: 57:10)



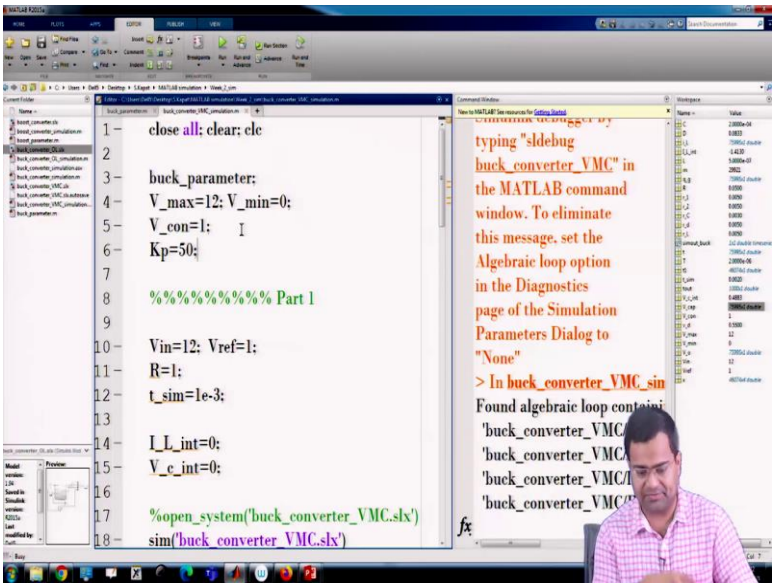
So, there we are connecting the output voltage directly and here, we are setting another constant which is call as a Vref. Now, so, there, we are setting Vref, this is my reference voltage now.

(Refer Slide Time: 57:49)



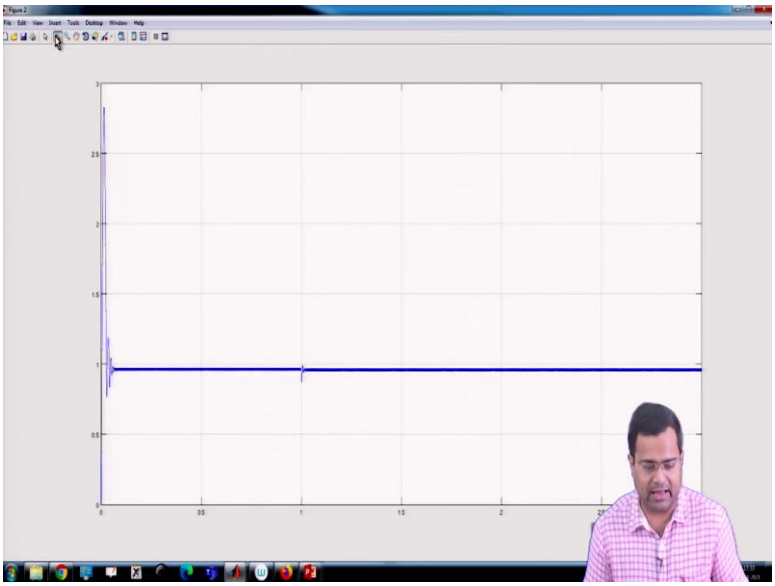
Now, this is my like a standard classical feedback control and we want to use a gain simply proportional control maybe you can use some gain of 100 so, I can gain something like Kp.

(Refer Slide Time: 57:53)

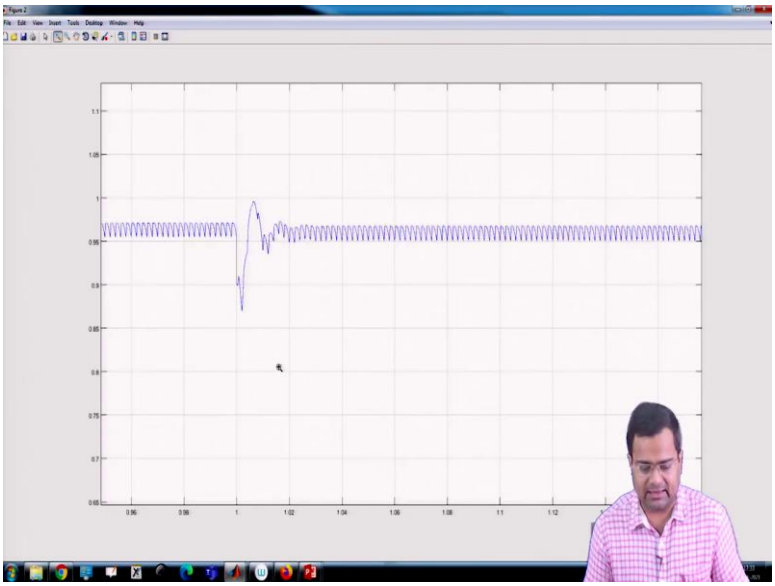


And this Kp I can now define like you know Kp equal to maybe 50. So, now, let us run it and see what happen.

(Refer Slide Time: 58:03)

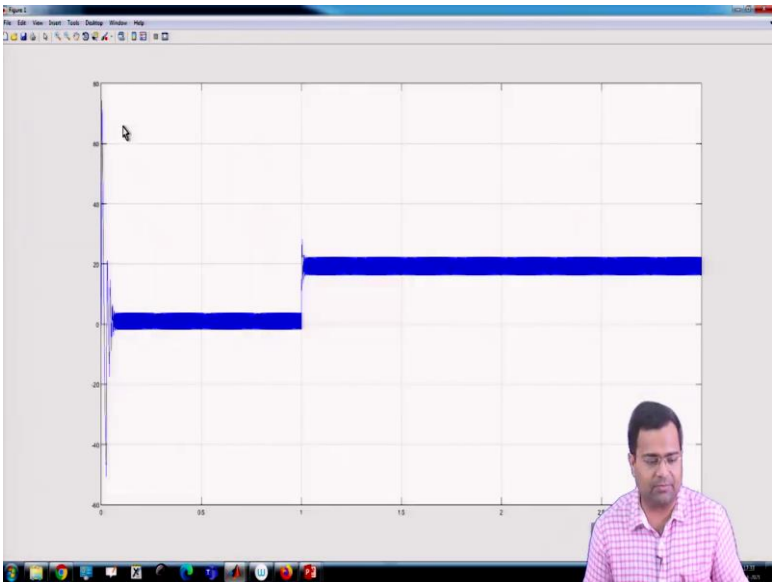


(Refer Slide Time: 58:09)



So, once we add proportional control, then we can see that the transient response is now the nature of transient response got changed.

(Refer Slide Time: 58:12)



(Refer Slide Time: 58:18)

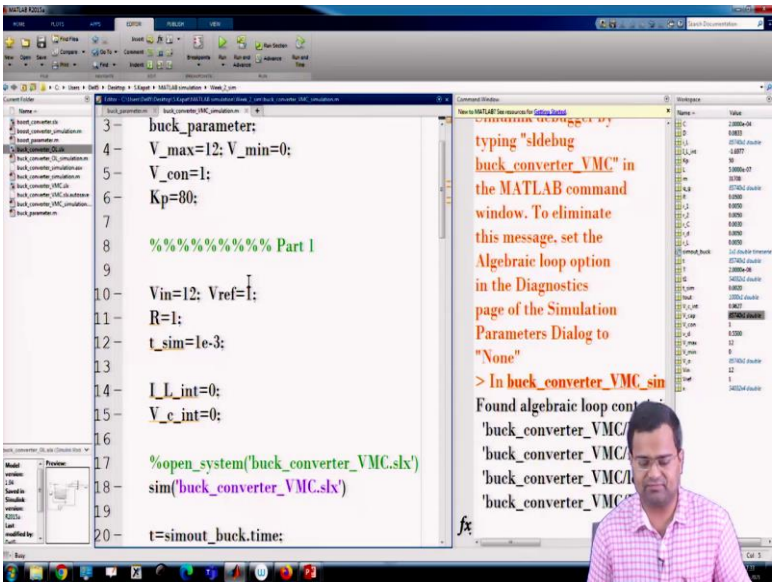
(Refer Slide Time: 58:20)

The nature of transient response got changed. So, it is a step of transient, but it is just a simple proportional control.

(Refer Slide Time: 58:30)

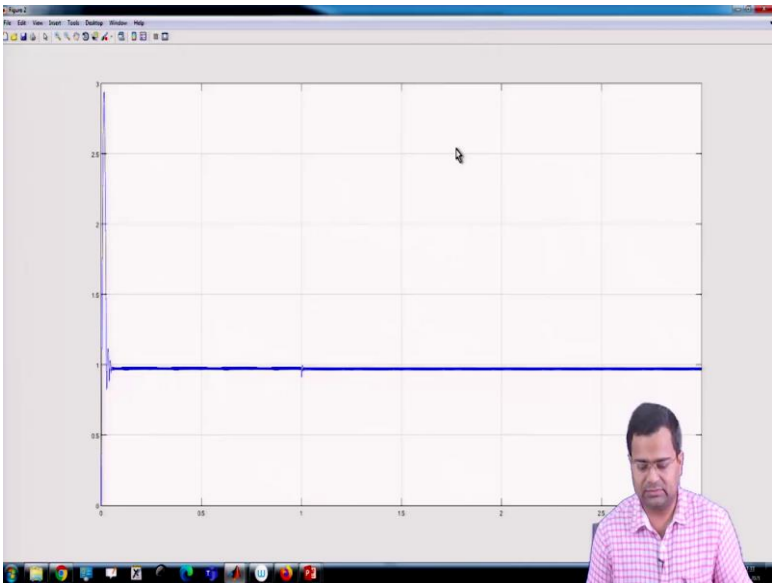
So, now, we can add more complex PID control and so on we can; we can do that.

(Refer Slide Time: 58:58)

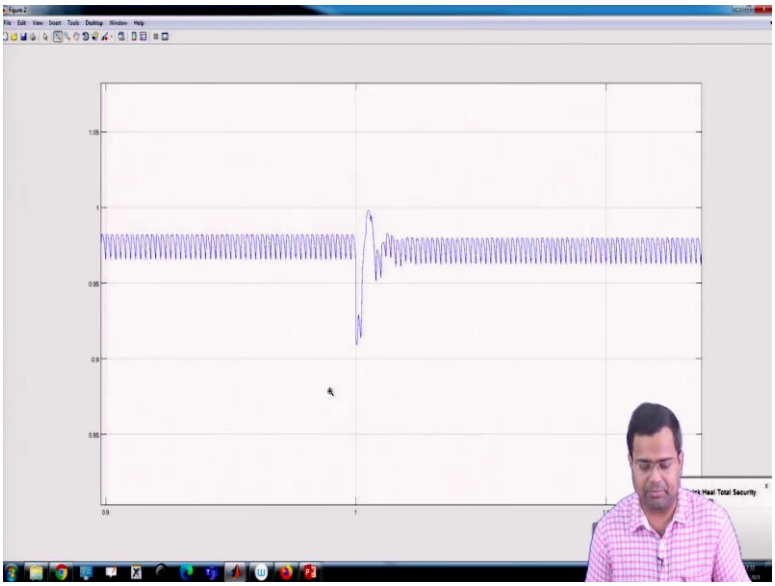


So, that means, we created a transient case study in the part 2, we have changed the load resistance and so, if we want to increase the proportional controller gain let us say 80, how does it look like?

(Refer Slide Time: 58:56)



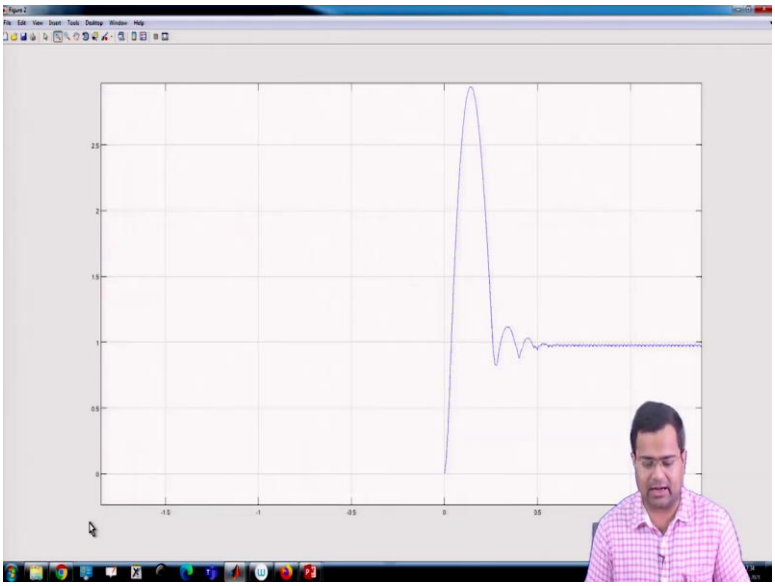
(Refer Slide Time: 59:01)



(Refer Slide Time: 59:03)

It has further improve the performance, but you know there are limit that there is a step; there is a steady state error in the output voltage.

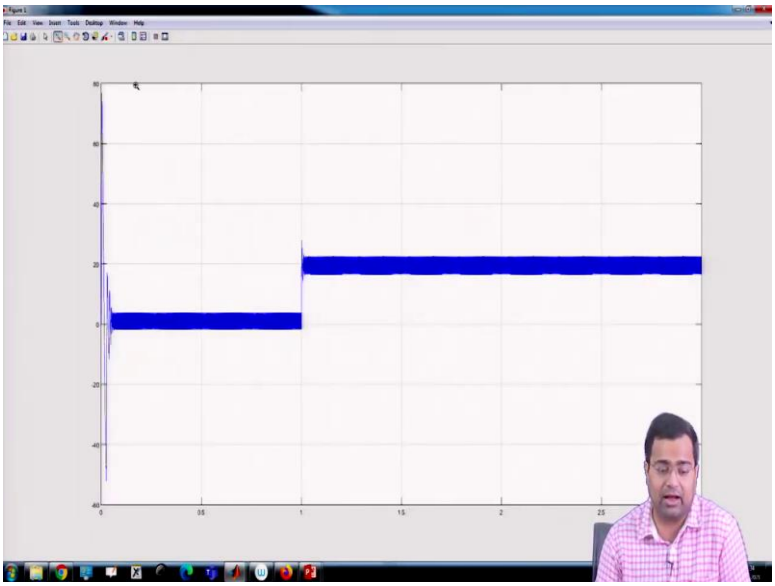
(Refer Slide Time: 59:16)



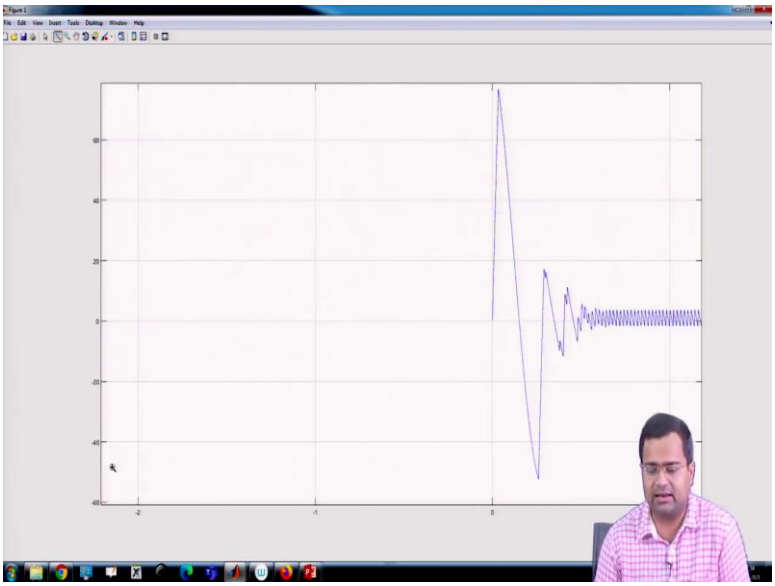
(Refer Slide Time: 59:18)

And you can also see during transient, the startup logic is something which is unacceptable, the output voltage reaches 3 volt which is not at all acceptable.

(Refer Slide Time: 59:23)

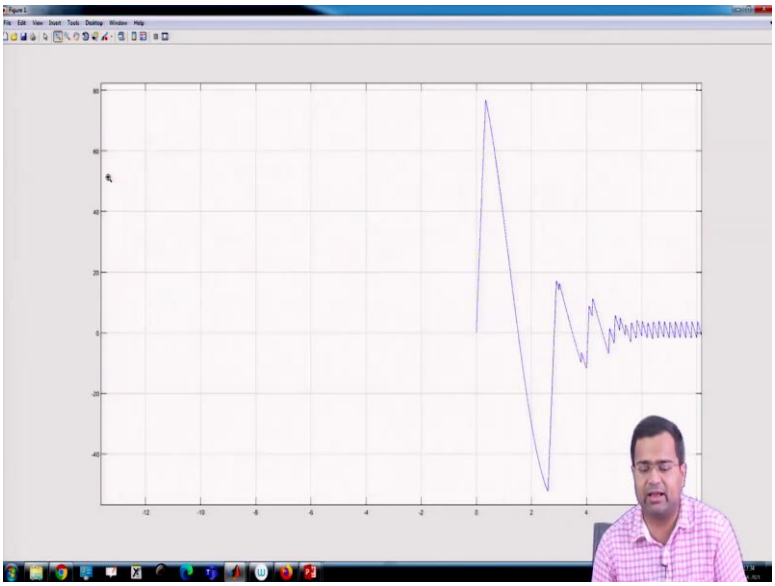


(Refer Slide Time: 59:30)



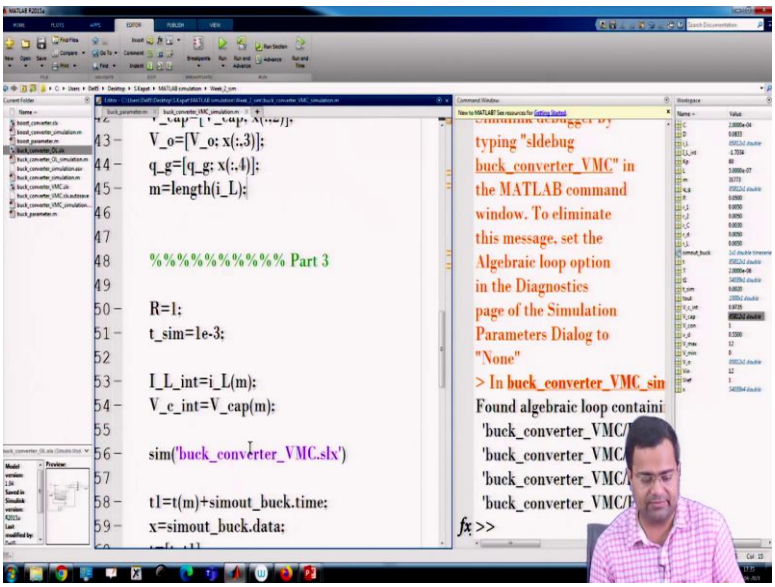
And if you see current, current is reaching 80 you know volt 80 ampere which is again like unacceptable.

(Refer Slide Time: 59:36)



So, this kind of we need to do something different in order to enable this startup operation ok, but at least we have learned how to run this simulation and we can do the same exercise for the boost converter also, but today I mean at in today class, I want to limit you know we can create such multiple transient case study we can create, and we can keep on adding.

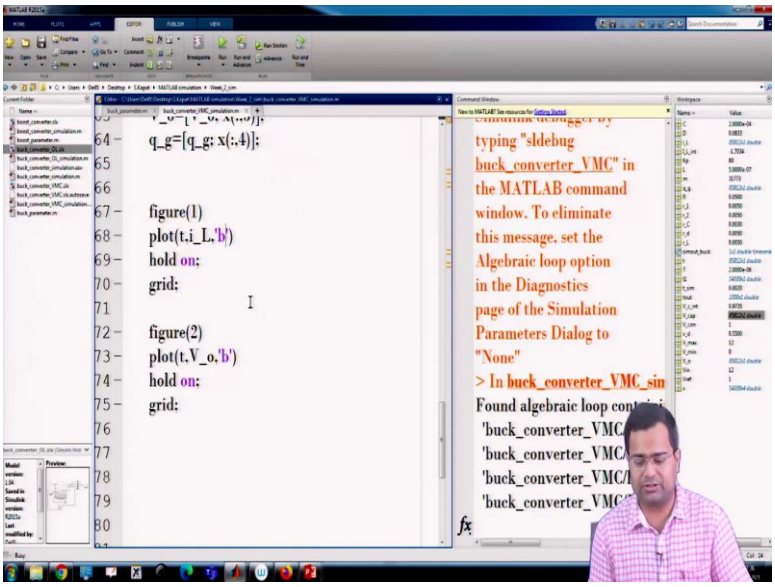
(Refer Slide Time: 60:10)



For example, you know I want to again go for third step, step down transient so, what I will do? Part 3. Now, we go back to let us say around 1 Ampere, 1 ohm maybe this time

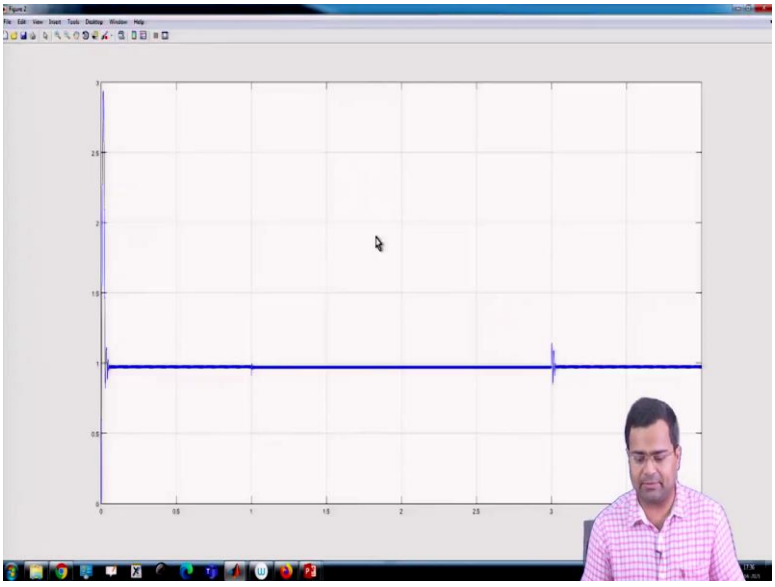
we change simulation time, it will again take, but now, we have to call what is my m and m is my again length of i L length and again, it will update all these value and again, it will take what was my previous t and accordingly, it will update and you know we want to plot after.

(Refer Slide Time: 60:59)



So, now, we want to plot like a this time like a green plot, we want to do green almost sorry this is the whole entire plot. So, let us see what happen if will do step up and step-down transient.

(Refer Slide Time: 61:09)



So, now, you can see the whole step up and step-down.

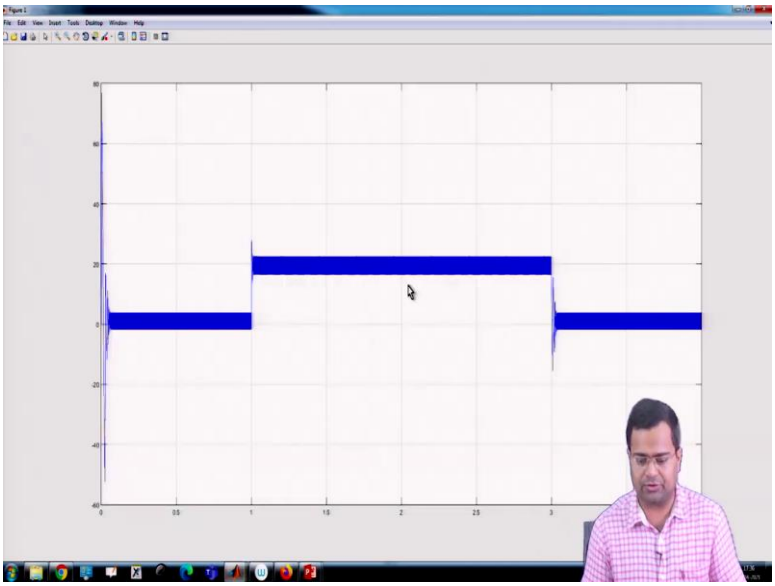
(Refer Slide Time: 61:14)

This was the step up transient.

(Refer Slide Time: 61:18)

This is the step-down transient, huge overshoot.

(Refer Slide Time: 61:21)



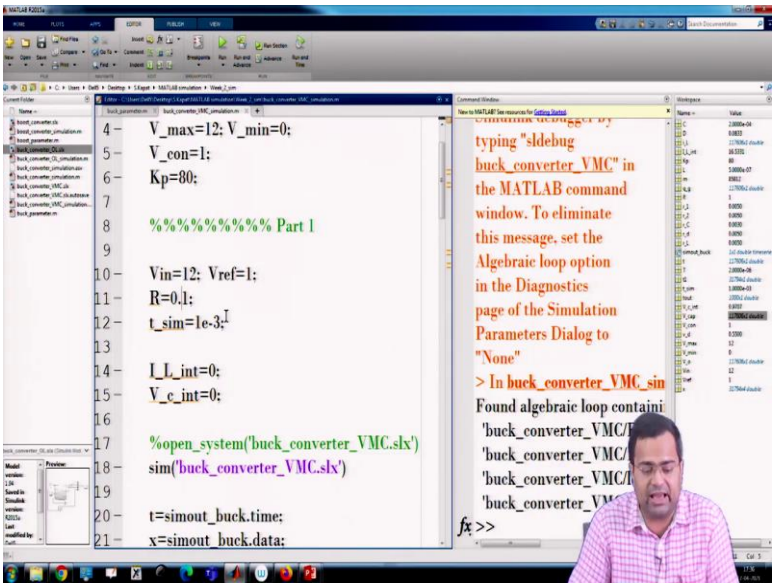
The current also undergoes like a step up and step-down transient.

(Refer Slide Time: 61:26)

(Refer Slide Time: 61:29)

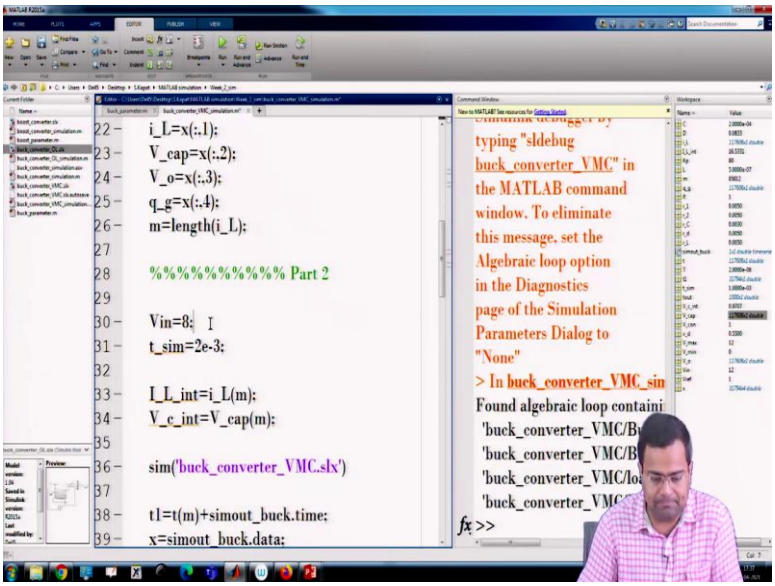
So, we can create such multiple test cases using Simulink, but we do not need to go physically, change the very well you know create adding multiple switch and all these block we can actually insert inside once you know subsystem, we do not want to do that.

(Refer Slide Time: 61:57)



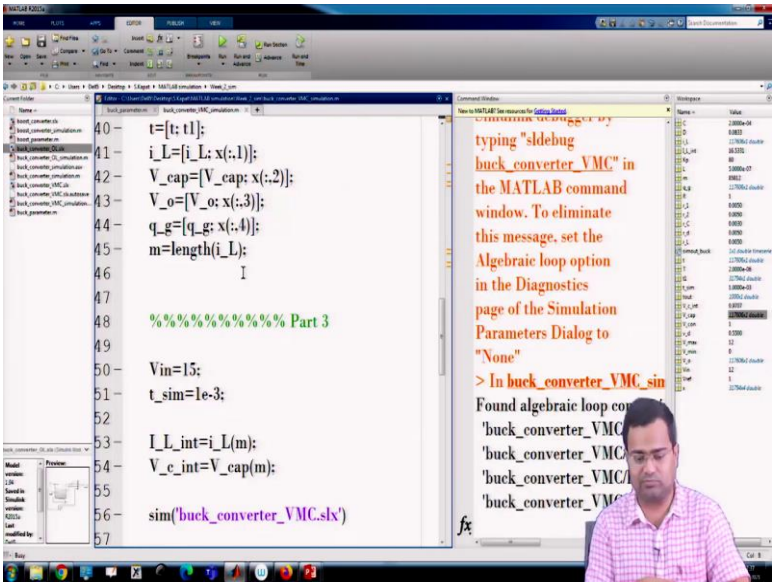
In fact, what we can do another case for example, we change the load, now we do not want to change the load, we keep R equal to let us say 0.1.

(Refer Slide Time: 62:07)



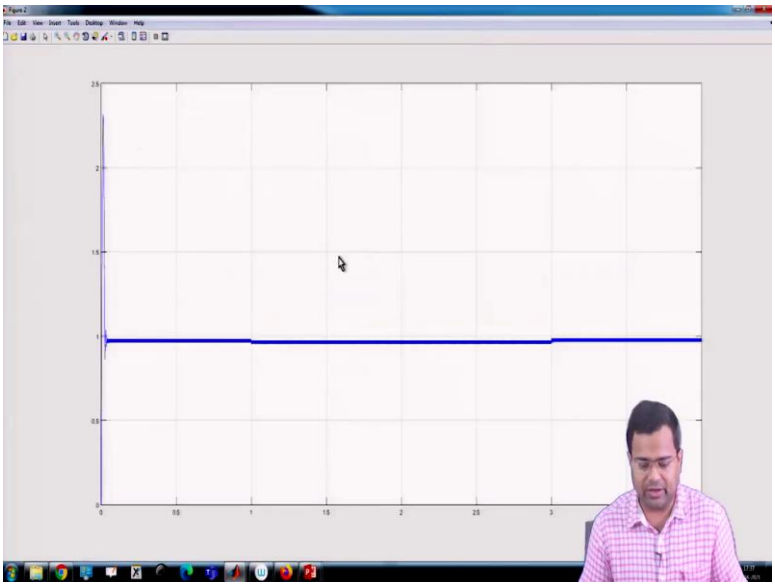
But we want to change the input voltage it was 12 volt now, we just want to change the Vin, Vin to 8 volt.

(Refer Slide Time: 62:13)



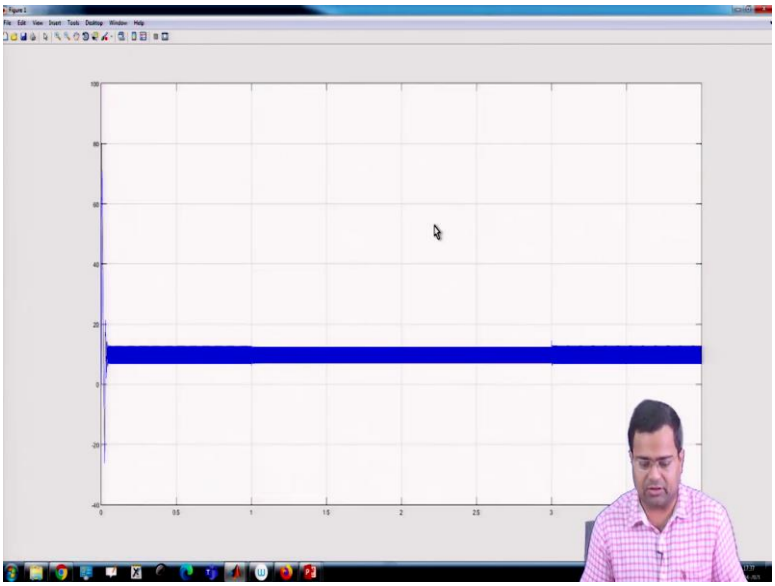
Again, Vin to 15 volt and run it and see what happen? Yes.

(Refer Slide Time: 62:21)



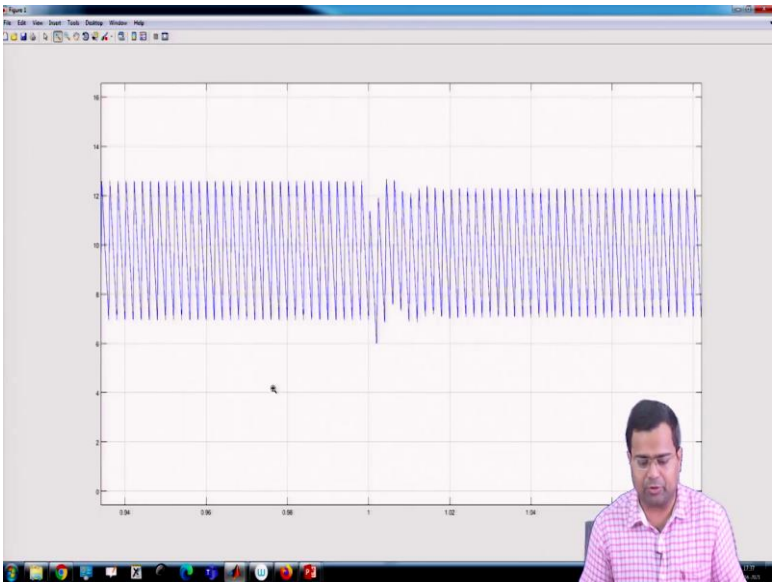
(Refer Slide Time: 62:26)

(Refer Slide Time: 62:30)



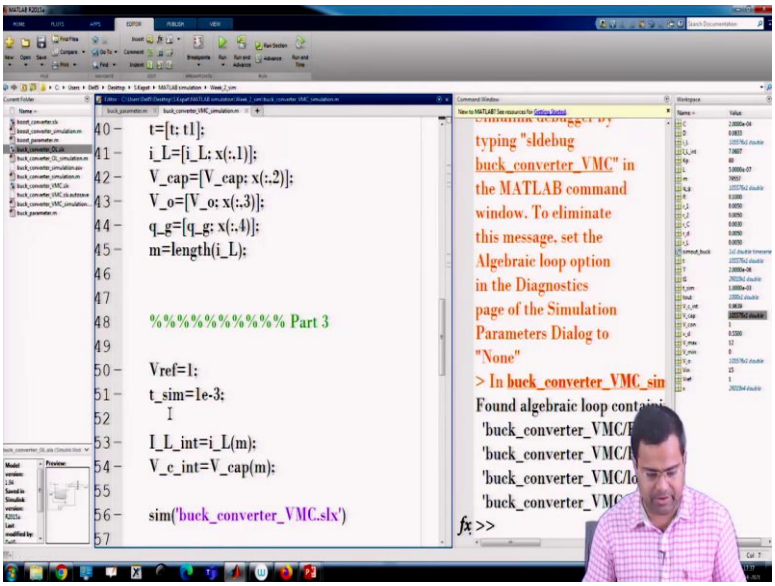
So, we can say there is not much effect, this is a supply transient we have made.

(Refer Slide Time: 62:34)



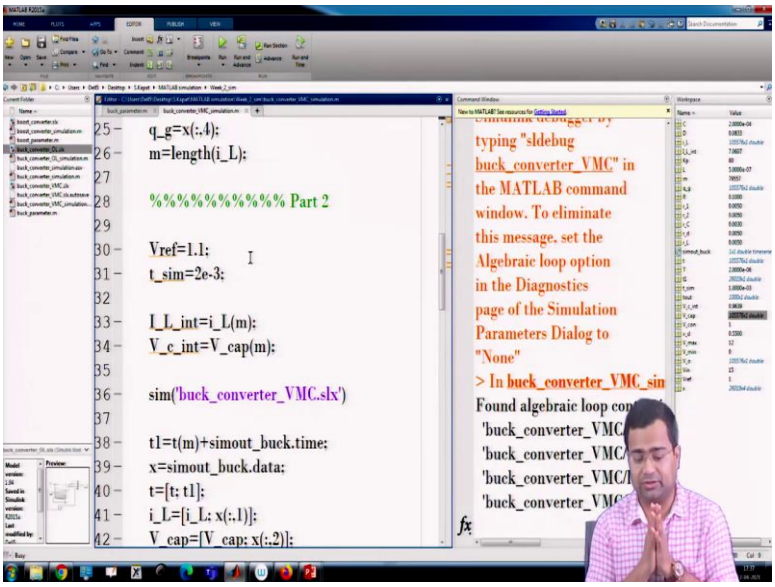
And current also has not changed drastically so, these are possible, all these you know you can run it.

(Refer Slide Time: 62:41)



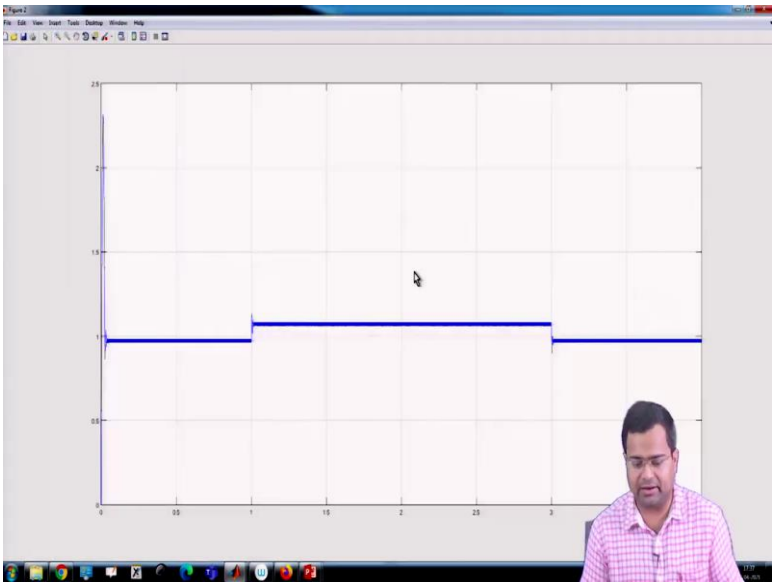
And the third one which probably we want to see, it will be 1 volt coming back.

(Refer Slide Time: 62:50)



I want to make Vref equal to 1.1 ok. So, and to see what happen? Yes.

(Refer Slide Time: 62:57)



So, we can create a reference voltage transient.

(Refer Slide Time: 63:02)

You know there are there is a huge overshoot and undershoot which is not acceptable.

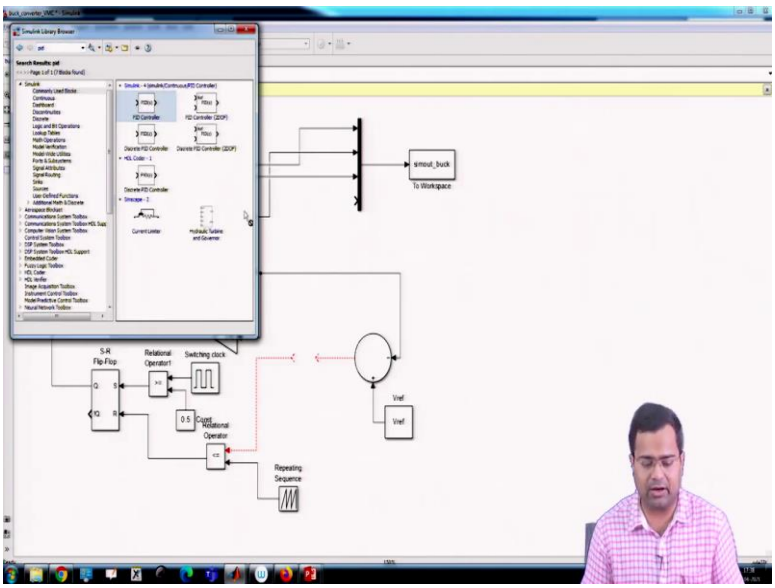
(Refer Slide Time: 63:07)

(Refer Slide Time: 63:10)

(Refer Slide Time: 63:15)

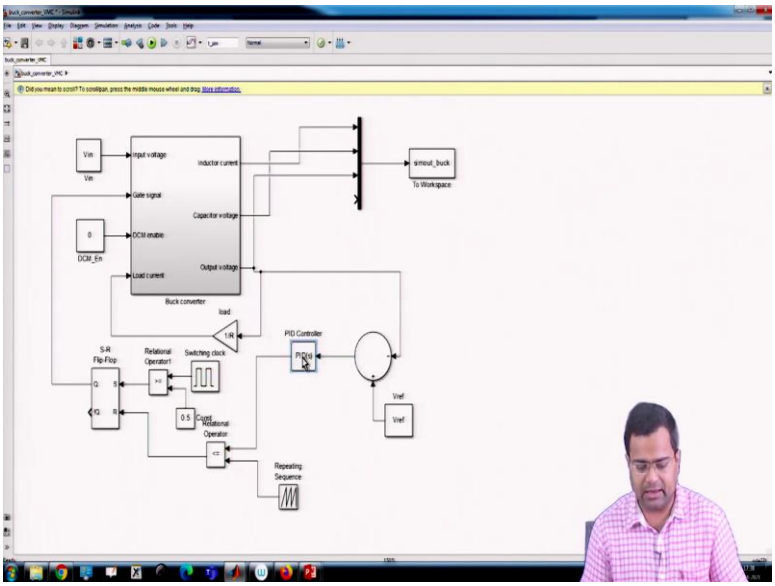
There can be huge you know current overshoot. So, we need to reduce the gain and we can check, but this is just proportional control.

(Refer Slide Time: 63:28)

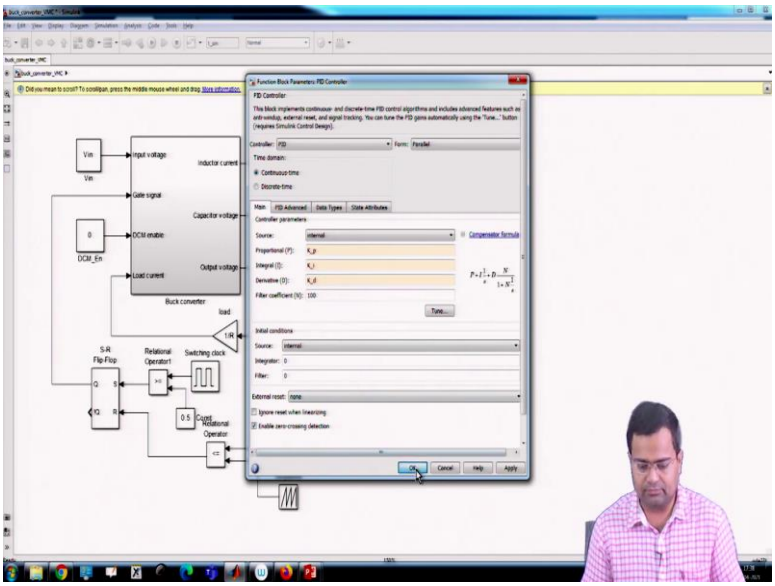


So, now, finally, we can consider a controller yeah, which is not simply proportional controller.

(Refer Slide Time: 63:42)

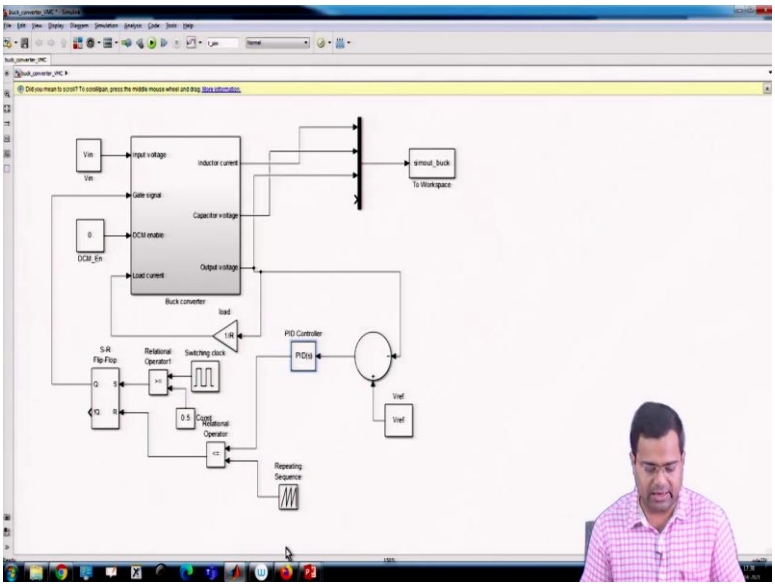


(Refer Slide Time: 63:47)

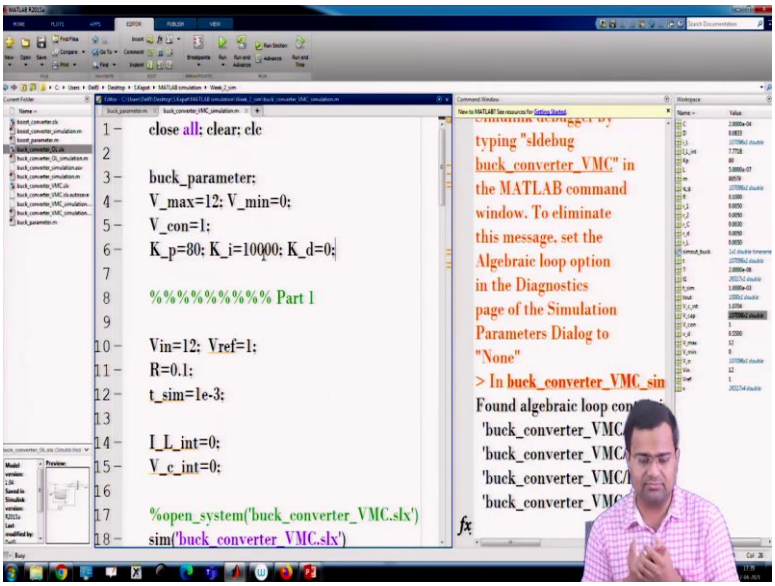


So, we can; we can consider let us say a PID controller, we can plug in a PID controller, we can use it all these PID controllers and here, you can use Kp I mean K i K p; K i, K d.

(Refer Slide Time: 64:01)

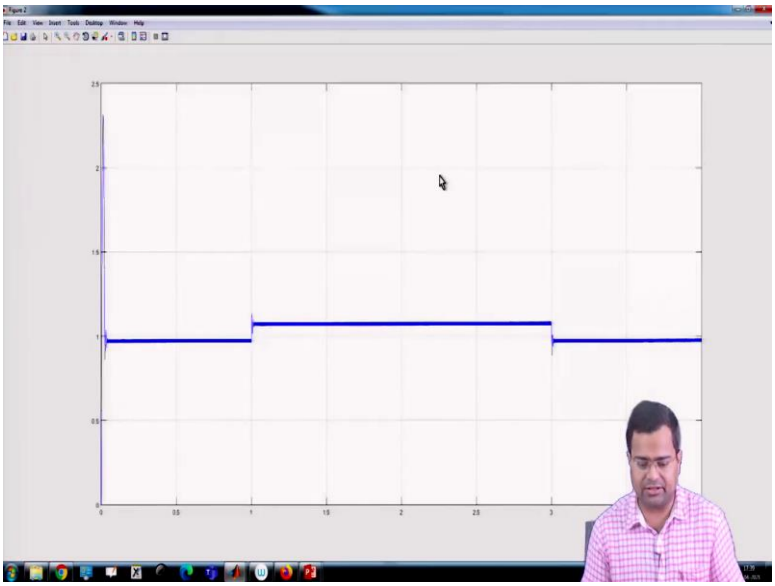


(Refer Slide Time: 64:06)



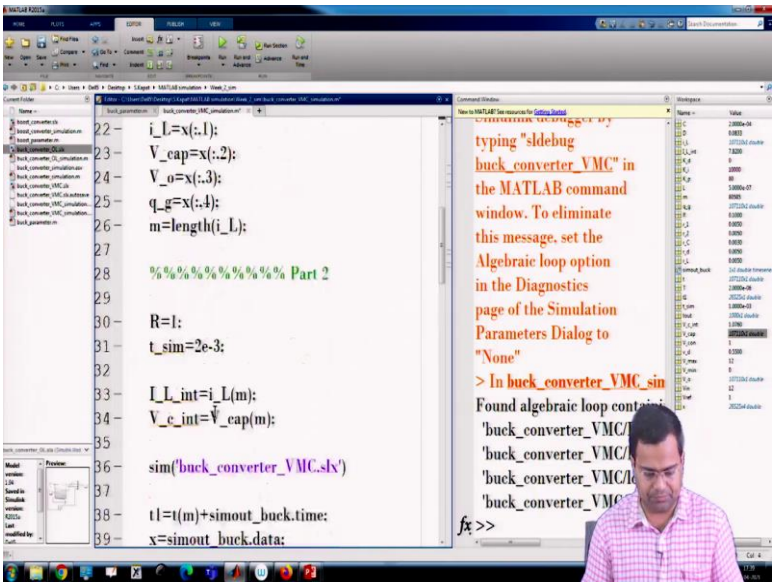
The controller again you can plug in here and you can accordingly create K p equal to this, then K i equal to maybe 10000 and initial we are taking K d equal to 0. Now, I can run it.

(Refer Slide Time: 64:22)



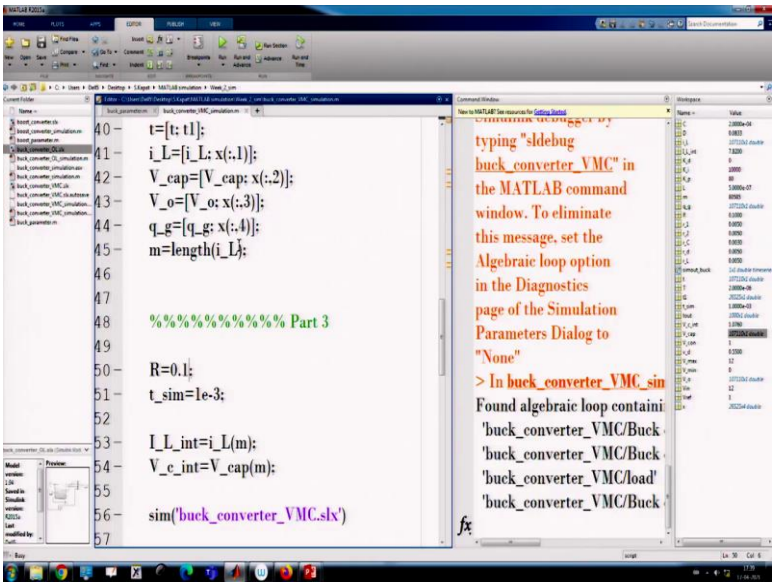
So, this we can create transient simulation case study.

(Refer Slide Time: 64:40)



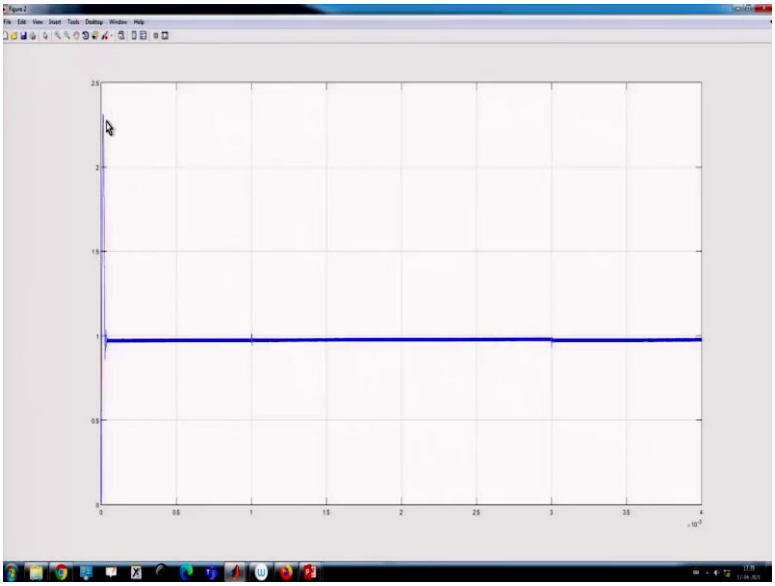
And you know here, we can again change the load resistance.

(Refer Slide Time: 64:45)



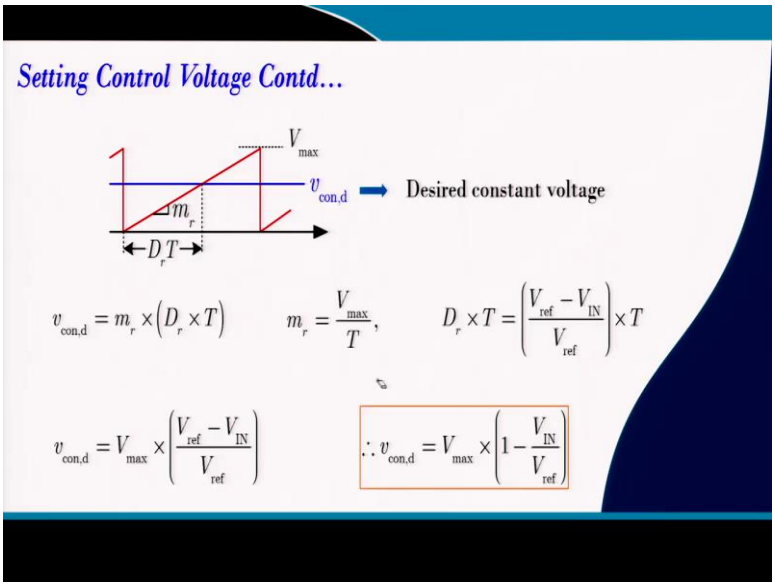
We want to see the load transient R equal to 1 ohm, R equal to and so on. So, we can see the controller effect yeah.

(Refer Slide Time: 64:52)



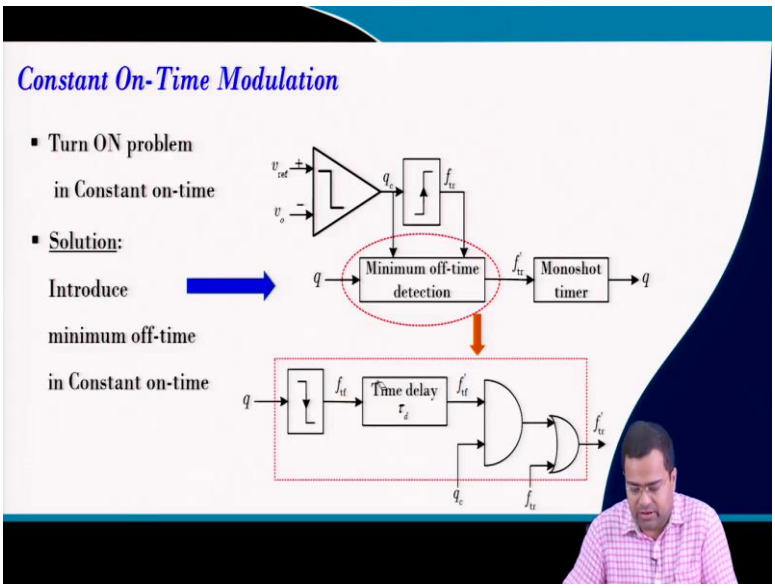
Load transient scenario step-up, step-down transient.

(Refer Slide Time: 65:04)



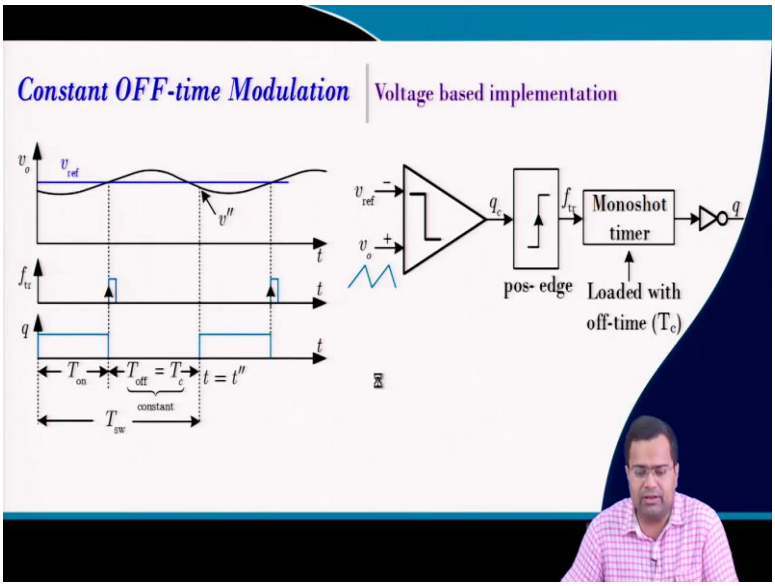
So, this actually we can implement various types of logic setting the control voltage, we can accordingly set the desired value that we have discussed earlier.

(Refer Slide Time: 65:09)

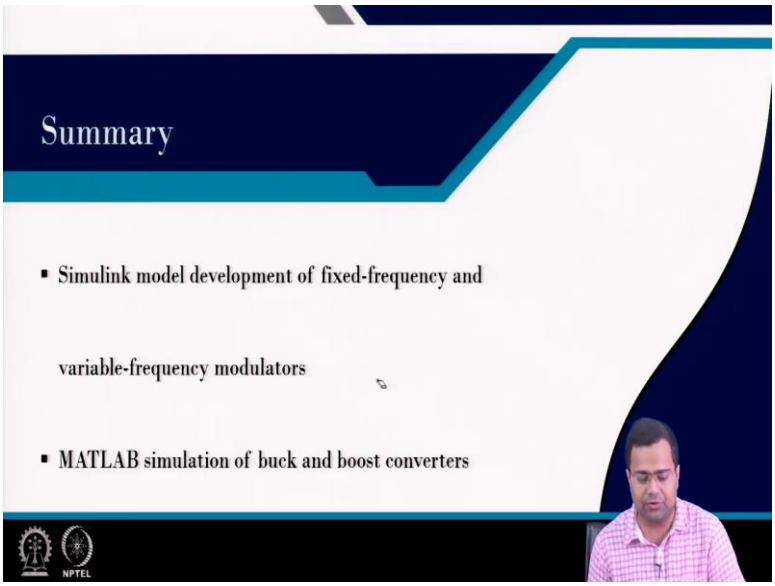


We can do constant on time control, but today, I am not going to demonstrate constant on-time because we will implement this constant on-time in the subsequent lecture, we will where we will have a dedicated you know variable frequency control method where we will have a constant on-time block and we will also use this for light load control. So, this block will be implemented in the subsequent lecture.

(Refer Slide Time: 65:39)



(Refer Slide Time: 65:42)



So, today we just wanted to show that the Simulink model development for fixed frequency, we have not yet gone to the variable frequency that we have kept it for the subsequent class and we have shown the interactive case study of a buck converter and we already have a built in model for boost.

So, we can do it that means, using custom MATLAB simulation, we can call the Simulink file, we do not have to run it through Simulink scope and you can directly run

through MATLAB file and you can speed up the simulation and you can create multiple transient case study. So, with this, we want to you know finish this lecture.

Thank you very much.