**Control and Tuning Methods in Switched Mode Power Converters**
Prof. Santanu Kapat
Department of Electrical Engineering
Indian Institute of Technology, Kharagpur

Module - 01
Switched Mode Power Converters and Simulation
Lecture - 05
Demonstration of MATLAB Simulation

Welcome. In this lecture I want to present MATLAB Demonstration of you know Simulink Model Development, as well as Simulation of Switch Mode Power Converter.

(Refer Slide Time: 00:41)
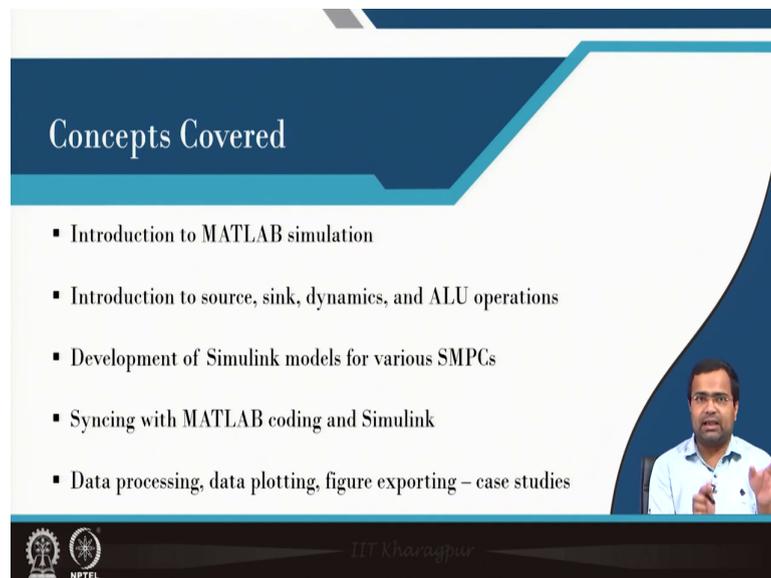
What are the topics that we are going to cover? First introduction to MATLAB simulation. Then, introduction to different blocks of Simulink like, source-sink dynamics and arithmetic and logic operation, then development of Simulink model for various switch mode power converters. Particularly, you know buck converter boost converter we did. In fact, we can develop, you know, isolated converter various types of converter that you can plug in into your Simulink.

So, we will develop from the scratch and whatever equation we have written, that we want to create the Simulink model for those mathematical block diagrams. Syncing then with

MATLAB coding and Simulink. That means, synchronizing MATLAB code and Simulink and finally, I will also show how to process data?

That means, if you generate if you see the scope simulation result, which does not look good, we need to take those data into the workspace and then from the work space we need to plot it, and we have to create according this axis command and then we will show some case study simulation.

(Refer Slide Time: 02:01)
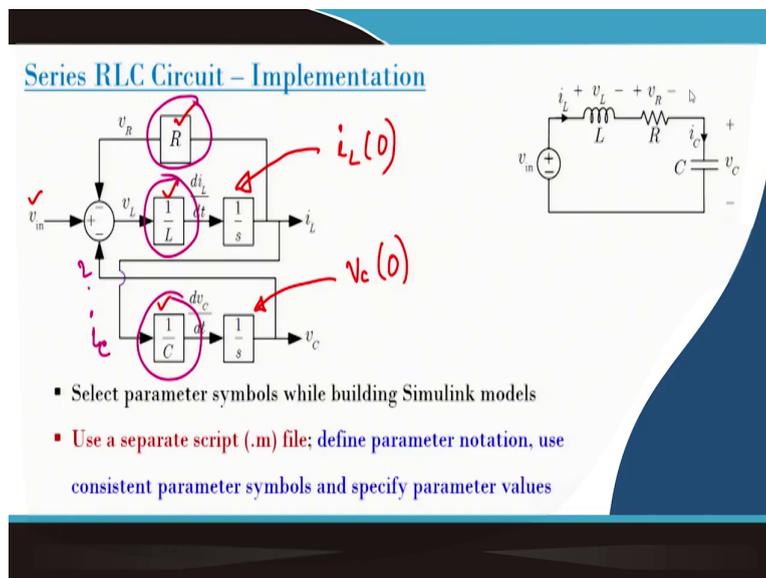
So, first circuit that we are going to consider is the series RLC circuit. In this circuit, we know we have in the previous lecture, you know we have already implemented that develop the model along with the block diagram. And, this one we are going to make the Simulink model for this particular series RLC circuit.

So, what is necessary? So, whenever we try to develop the Simulink model, you know there are certain parameters. For example, we have the resistance; we have this 1 by L, we need 1 by C, we also need the source, then we need to set earlier, we need to set the initial condition of the current here and we need to set the initial condition of the voltage here. So, all these parts we need to plug in we need to put this information in order to run the simulation.

And, in the Simulink model, we will plug in this parameter instead of putting value. So, that we can change the parameter separately using a script file. So, what we have to do? We need

to create a separate script file, which is a dot m file where we will define all the parameter the notation that we are using here. And, we need to use a consistent parameter symbol.

Otherwise, it will give error that means whatever symbol we will use in the Simulink, the same symbol we have to define in the dot m file. And, then we need to specify the value of that particular let us say for the inductance value, we need to give what is the value of the inductor and what is the value of the capacitor? So, with this, we are going to start.

(Refer Slide Time: 03:48)

Now, in the first one I will tell you dot m file, I will go to that. So, suppose I choose an inductance and if I do not specify, these are the command file. So, it does not matter it will not be executed, but these are the file which will be executed. So, when we in the MATLAB we will only give values. So, values mean inductors it will be considered in Henry capacitor in Farad resistance in ohm, and current in ampere and voltage in a volt. And, if we want 2 micro Henry inductor so; that means, we have chosen here L equal to 2 micro Henry.

So, this can be written as 2 into 10 to the power minus 6 Henry and in MATLAB we will simply write 2 e minus 6 and we do not need to give a; that means it is Henry. So, we need to just specify the values here. So, this is the inductance value. Similarly, for this one capacitor, this is like 100 micro Farad capacitor. And, here we are considering load resistance to the 1 ohm.

So, we can choose any other resistance value for the simulation. Now, this block actually we know this is the parameter block. So, this has to be created as a dot m file, script file, and which extension file is simply dot m file. Now, let us go to the Simulink block and we will see how do we implement this block?

(Refer Slide Time: 05:35)
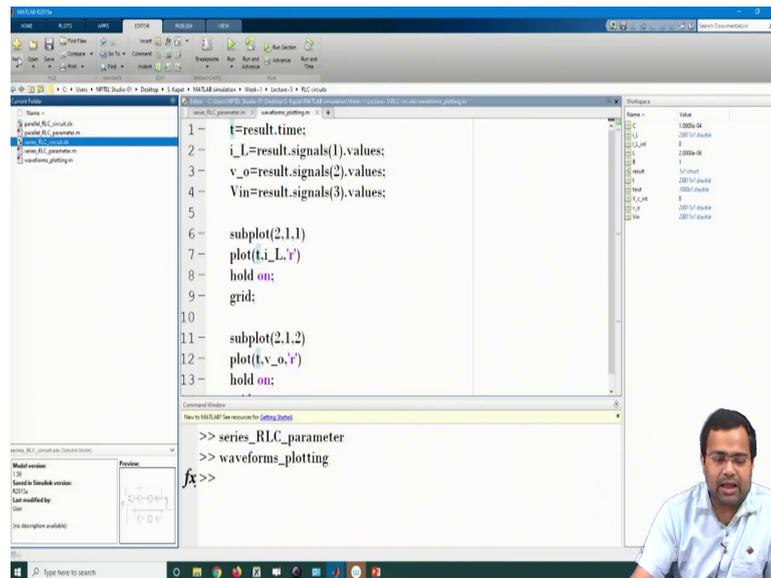
So, here initially I will open whenever I will open the MATLAB block simulation. You need to set the specific location, where you want to keep your simulation file. So, here this is the path that I have created for this particular case study and do not bother about this file. So, here I need to create a new script file. This is the new script file.

(Refer Slide Time: 06:04)
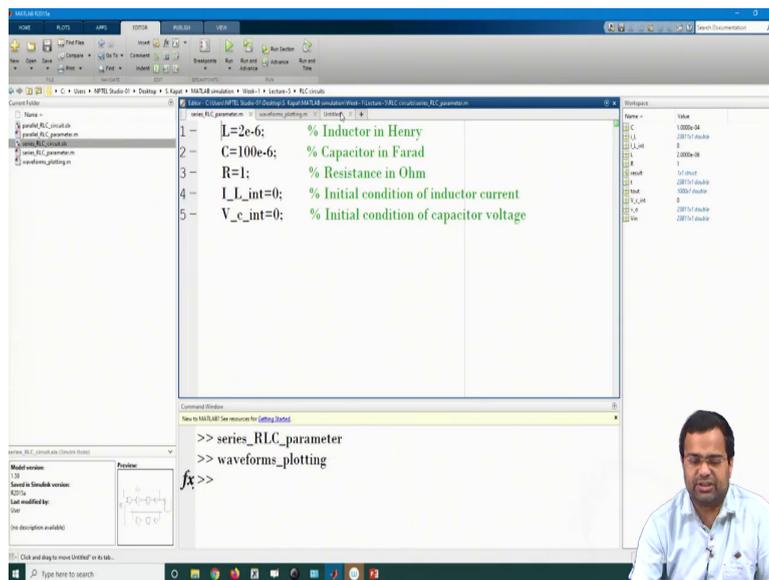
And, let us say I put an inductance value L equal to 2e minus 6. That means, this is a, inductance is equal to 2 micro Henry. I want to use a capacitor to be let us say 100e minus 6, this is 100 micro Farad.

Then, I want to use a resistance in this case equal to 1 ohm. Since we are not putting any symbol, it will be treated as L into L in terms of Henry C in terms of capacitor in Farad and R in terms of resistance in ohm. So, the same thing I can say, so, I have created a file just to give an intuition ok.

(Refer Slide Time: 06:53)
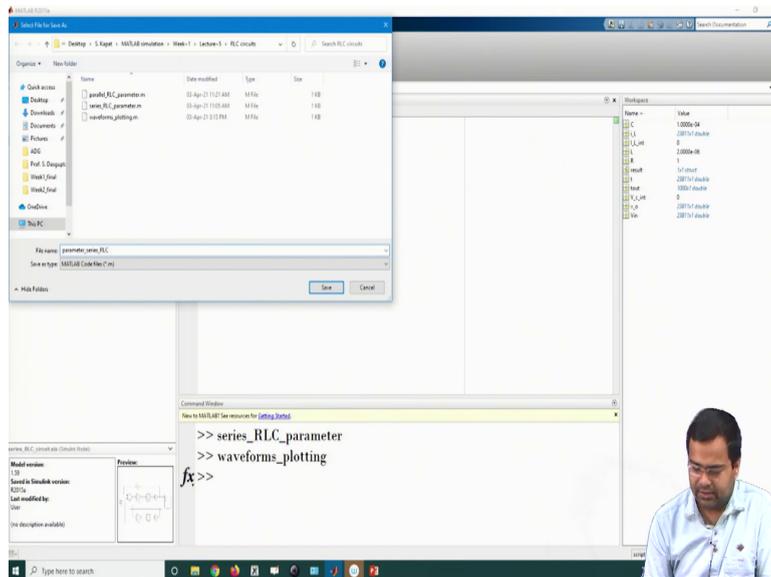
Now, I need to save this file somewhere. Let us say this is the parameter file. Parameter for the series RLC, you can define your own parameter file. But, these three are not sufficient, because what else is needed in our simulation. We also need the initial values of this and we discuss we need to define the initial value of the inductor current and the initial value of the capacitor voltage.
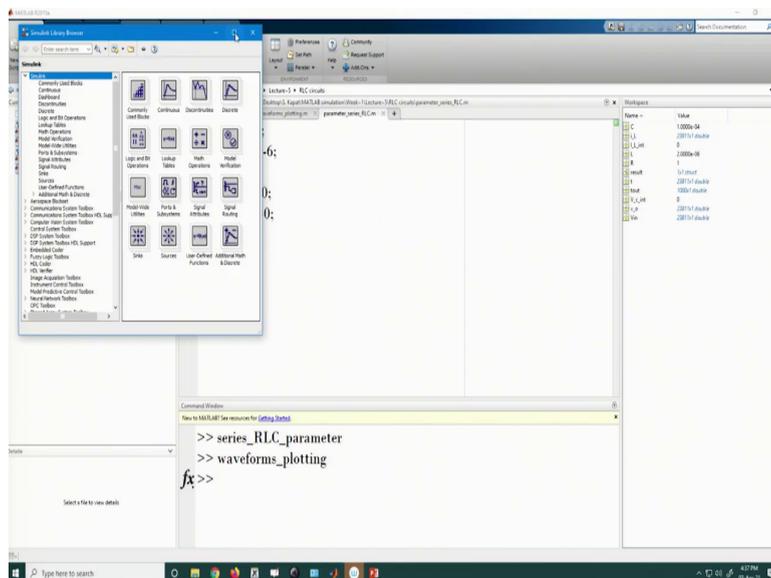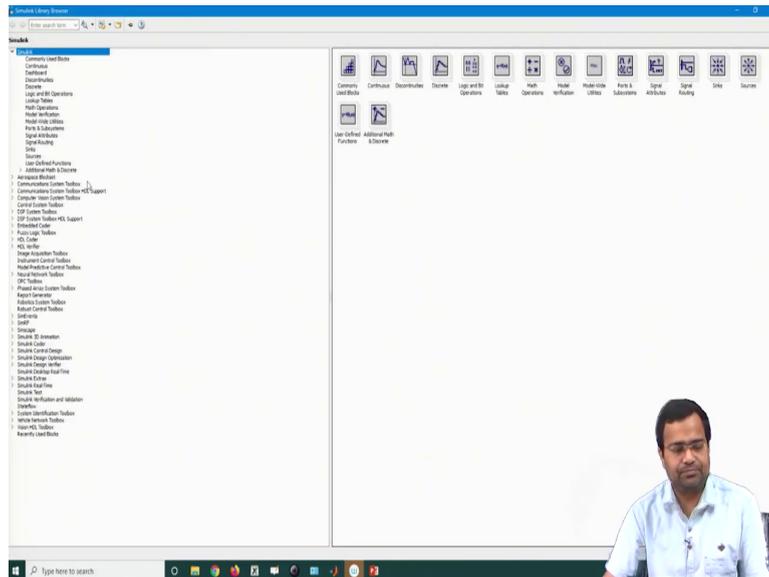
(Refer Slide Time: 07:33)
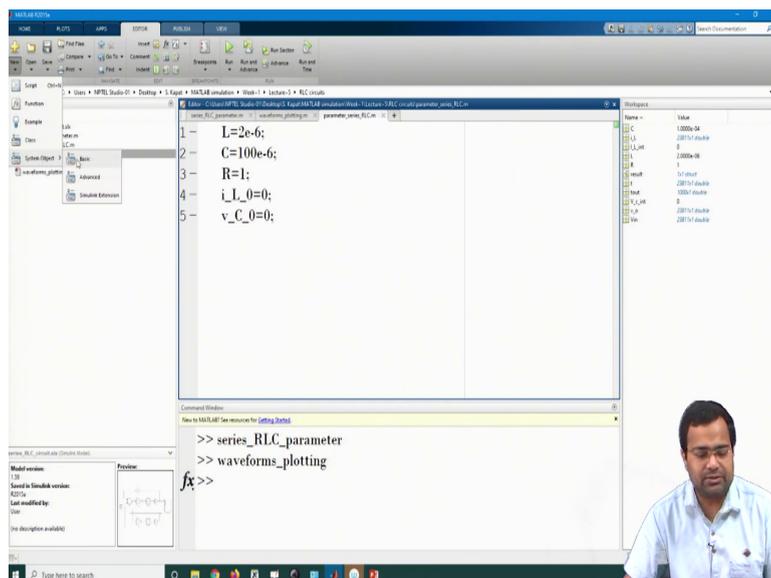
So, this initial value, we can initialize like we are talking about i L 0. This can be treated as initial value. Let us say this we take directly as a 0. Similarly, I need to take capacitor initial value. You can define your own initial notation, but it has to be consistent 0 ok. Now, this actually generates all these simulation files. And, we need to keep this simulation file in the backup and we have to run it.

Now, what is our next task? So, we need to develop a Simulink model ok. So, we need to create a new file. So, you need to create a Simulink file. Let me check where is the Simulink block here this is the Simulink. So, it will go to the home page of the MATLAB, then you will see the Simulink tab. So, it depends upon the version of the MATLAB that you are using and if we open then it should open a Simulink file ok.
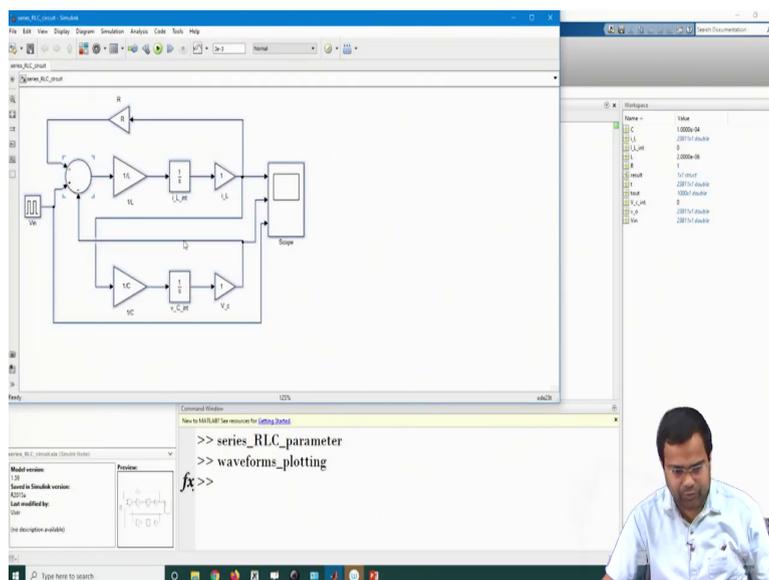
(Refer Slide Time: 09:04)

(Refer Slide Time: 09:09)

(Refer Slide Time: 09:17)

So, if now it you open a new simulation file, this is the Simulink library. Let us open a simulation file. So, I am just removing all these blocks from here. Created this is a simulation file, new Simulink file and in this Simulink file we want to build up the model that we discussed.

So, if we go to the previous simulation in this model, first we need to create this inductor current block, then we need to create the capacitor voltage on that path and then we have to combine this whole equation. Let us go and create.

(Refer Slide Time: 10:05)
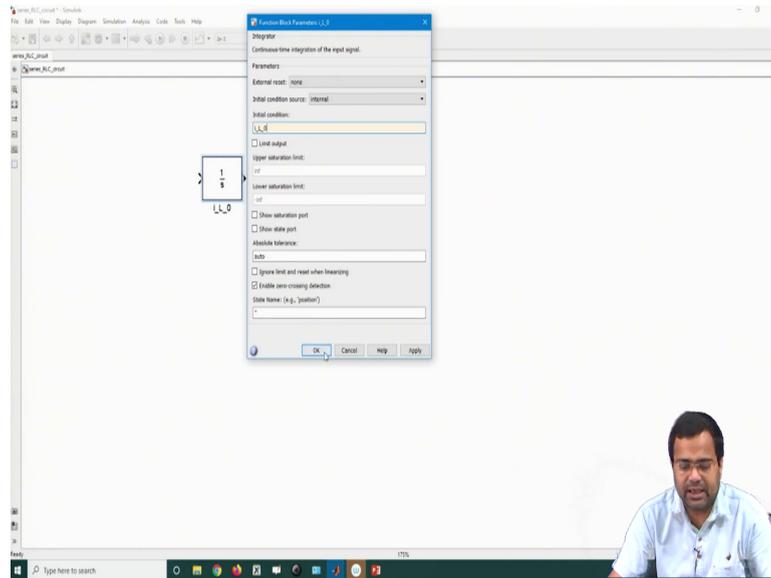
So, in this simulation if you go to library, there are commonly used block. And, we will use most of the blocks from here commonly used. The first block we need is the integrator; because this block is needed. What is the purpose of this block? This block is to generate the current and voltage just a minute.

So, this is my integrator block, and this block will set the initial condition. Here, the initial condition that we take must be consistent. So, this we should copy from the dot m file and you set it here as a dot m file. So, the same must be used.

Now, we want to generate another such block. We need to copy and paste control c and control v and here it is v C 0, because this is and again we can go and use this initial block and we should use this block.
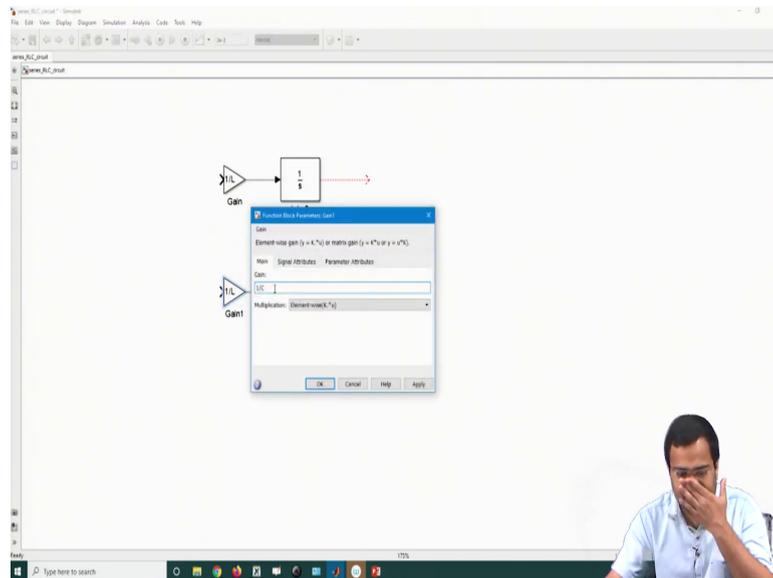
(Refer Slide Time: 11:11)
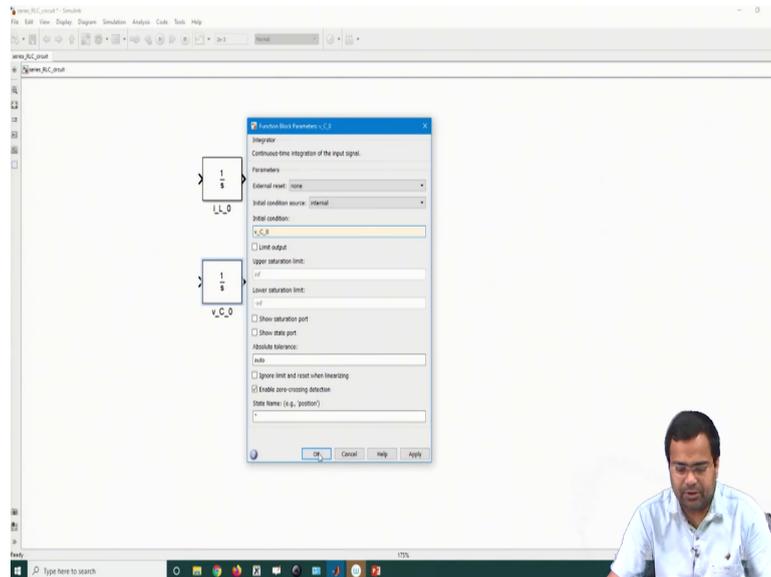
Now, we have place 2, one for inductor current another for capacitor voltage. What is next? So, that means the right side of this block is the inductor current and the left side of this block is d i L d t right. And, what is d i L d t? We have d i L d t, if we multiplied with 1 by L, then we will get v L inductor voltage. So, we need another block ok, Simulink library. So, here we need something like a gain block gain.

(Refer Slide Time: 12:01)
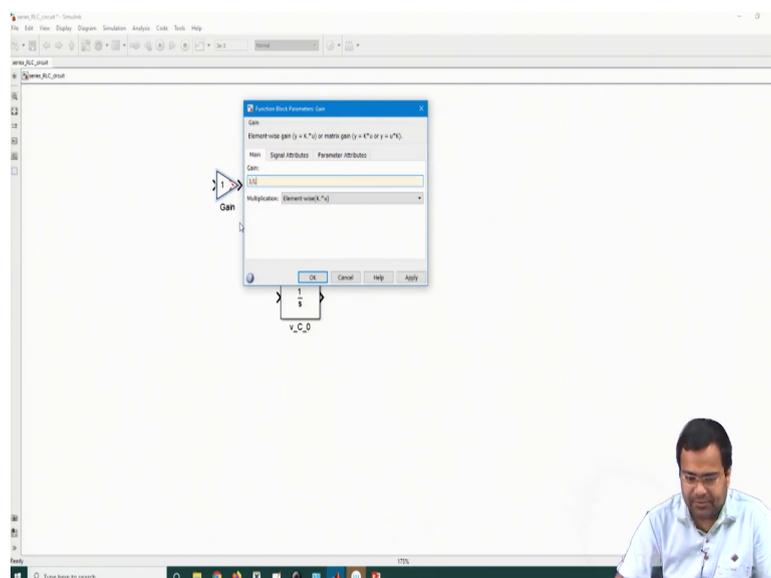
So, gain block this gain block will set 1 by L, 1 by L and this 1 by L, now it is a parameter. Similarly, we can copy paste this block and this block is used for 1 by C for the capacitor.

(Refer Slide Time: 12:20)

And, I am just referring to our presentation; that means this is the 1 by L block, because I am talking about this 1 by L block, and this is 1 by C block it is multiplied by 1 by S, that is my integrator, ok.

(Refer Slide Time: 12:56)
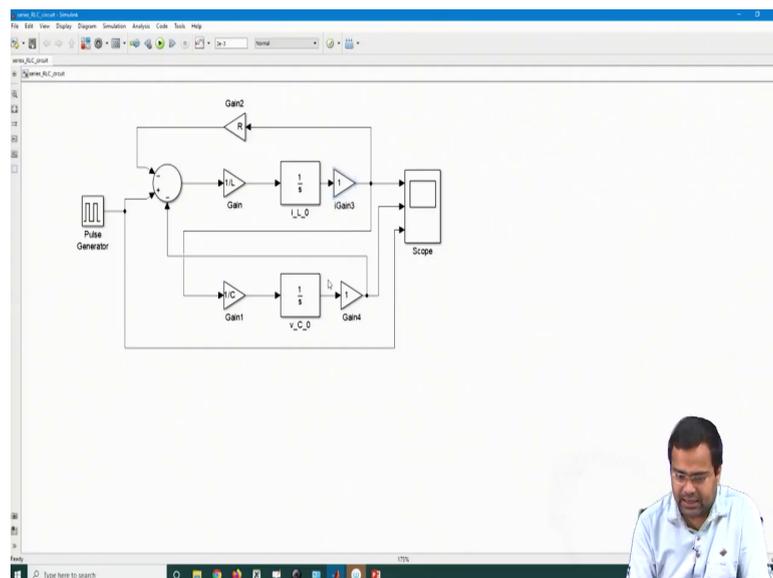
Next part that we need to use an adder block; that means, go back one summing block, one summing block should be we needed. And, this summing block what we need to add?

You know, for this series RLC circuit; we need to the side of this, this side of this terminal is my inductance voltage. For a series RLC circuit, what is the inductor voltage? That is because it is the input voltage, minus the voltage across resistance, minus the capacitor voltage; that means, we need plus minus, ok.

(Refer Slide Time: 13:31)
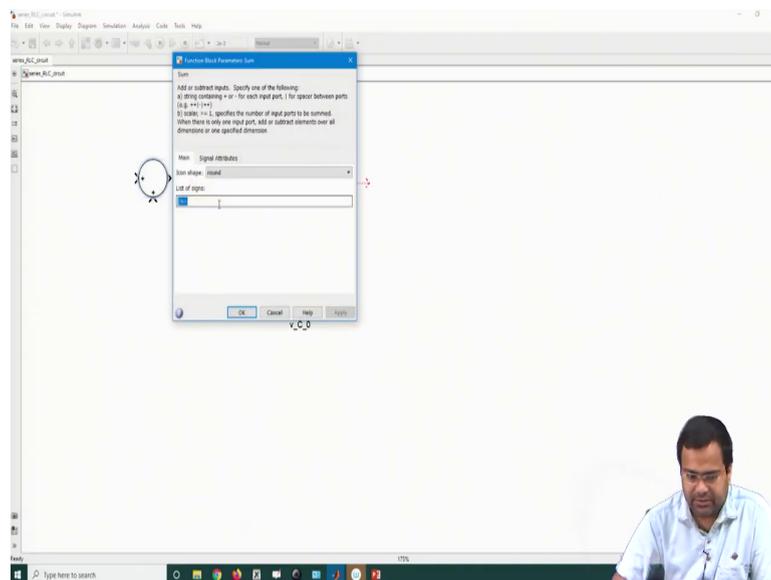
So, since we want to connect minus plus minus ok, this is for just notational simplification we join this block. Now, again, we want to create a block. If we rotate this block is resistance block, this we will use with this block and it will be this. So, what is this? If we go back to our presentation, this is my resistance block. So, I am talking about this block, which will be multiplied with the current and subtracted from the, this it will be subtracted ok.

Similarly, we need to subtract the capacitor voltage; that means, I am talking about this terminal, this particular terminal. So, once I go back then we need to subtract this voltage, ok alright. That means, we now then what is left? What is left is there, my supply ok. So, I can take a source. If you go to source, there are various types of sources. In the Simulink library, if you go to source there are various types of source.

So, let us say I choose a pulse generator; that means, I want to give at DC for short time then 0. So, repeated pulses you can generate any other signal also you can generate a sin wave AC, you can generate DC, even you can put a custom blocks for simulation, that is also possible. So, we will slowly move into more complex block, but we will start with a simple block.

(Refer Slide Time: 15:30)
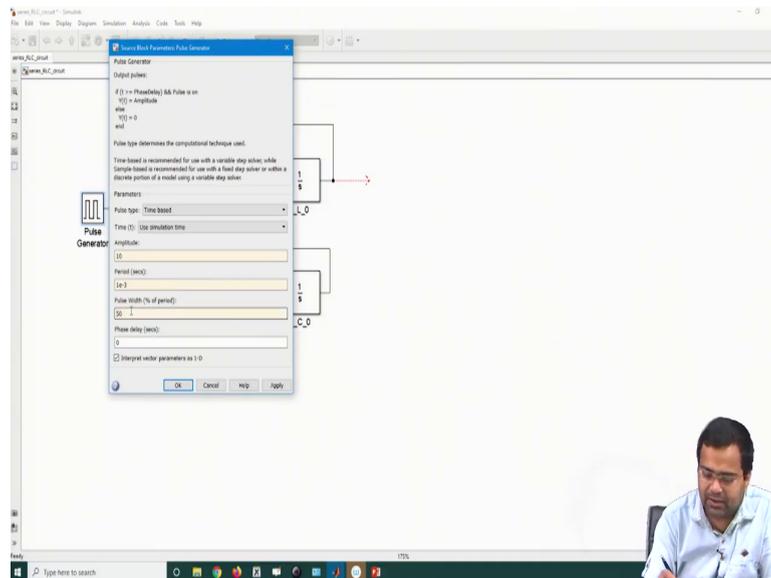
So, in this block we want to apply a 10 volt DC amplitude of 10 volt with a time period of 1 milli second 1e minus 3 and 50 percent with ratio; that means, we want to apply what we want to apply? That means, we want to apply a pulse, which width is 50 percent. And, during 50 percent time, the signal the voltage will be 10 volt, then remaining 50 percent time it will be 0 volt.

So, it will vary from 10 to 0 and with a time period of 1 millisecond that we say ok. The next plot if we go back is the capacitor voltage, which is if we multiply by 1 by C that is already there, that is my capacitor current. So, this terminal is my capacitor current. So, this is my capacitor current and in this case the capacitor current is same as the inductor current, because it is flowing through this part only series part.

So, in the Simulink I will simply connect this block with this block. With this in this block, simply connect this block.

(Refer Slide Time: 17:00)
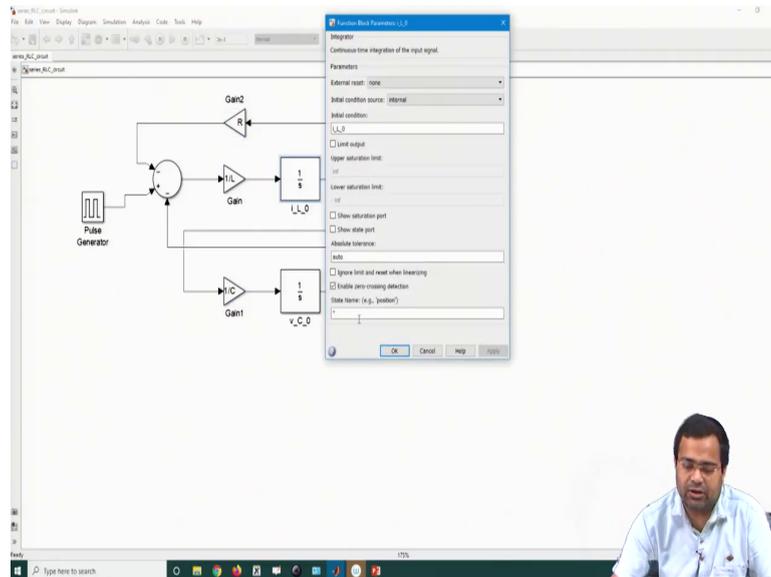
That means, now, we have defined 1 by L we have defined 1 by C, we have defined resistance; we have defined initial condition; we have defined this initial condition, ok. And, here we want to set the simulation for maybe 2 microsecond, 2 millisecond sorry, but we want to display the waveform.

In order to display waveform we have to go back and sink block, where you will find a scope. There are various types of scope. One scope is here; let us see the normal scope like which will show the time domain signal.

(Refer Slide Time: 17:36)
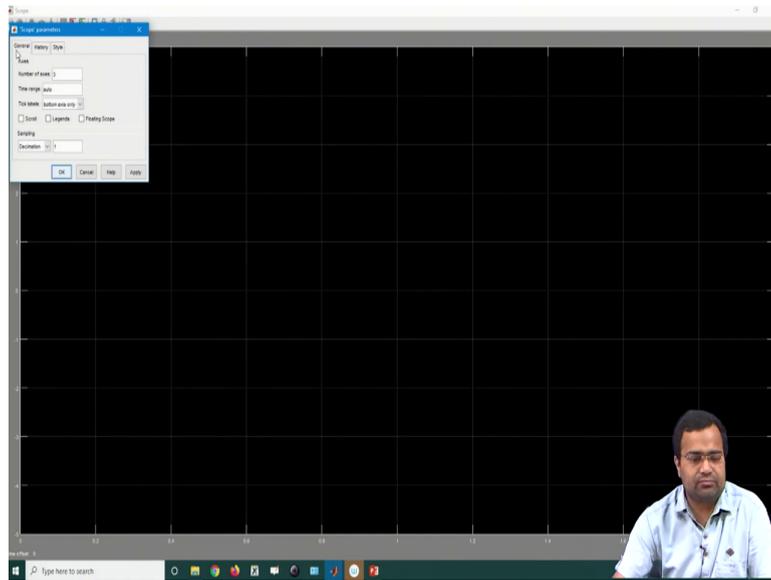
If we double click on the scope and go back to the scope in the setting block. Here you will get a setting block, where you can select 2 axes even you can select 3 axes whatever you want. And, on that axis you know I think on this axis so, this is my axis.

(Refer Slide Time: 18:08)
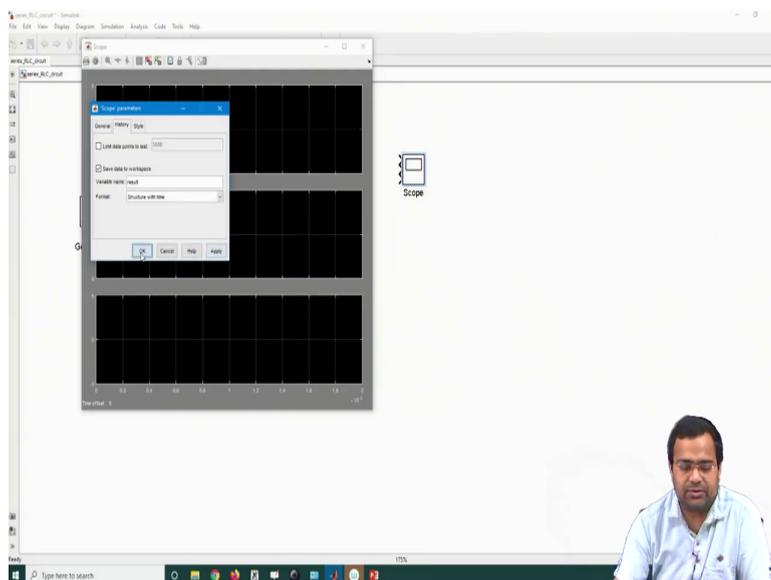
I am just considering 2 axes history term. I do not want to limit the data. Because, history term is that the final 5000 data point it will display, but that we do not want. I want to; I want to show the complete simulation. Because, sometime this history block help in order to

because if you want to display each and every data, then the simulation can take long time longer duration.

But, if we want to speed up simulation we want to show some final data point. So, this is used for limiting data, but here we are not using, we are not limiting data. But what we want? We want to save the data result scope result into work space.

So, I have to select this work space and I can define the custom. Let us say I am just defining something called result, result block and I will structure with time, structure with time. So, it will show something like this scope.

(Refer Slide Time: 19:10)
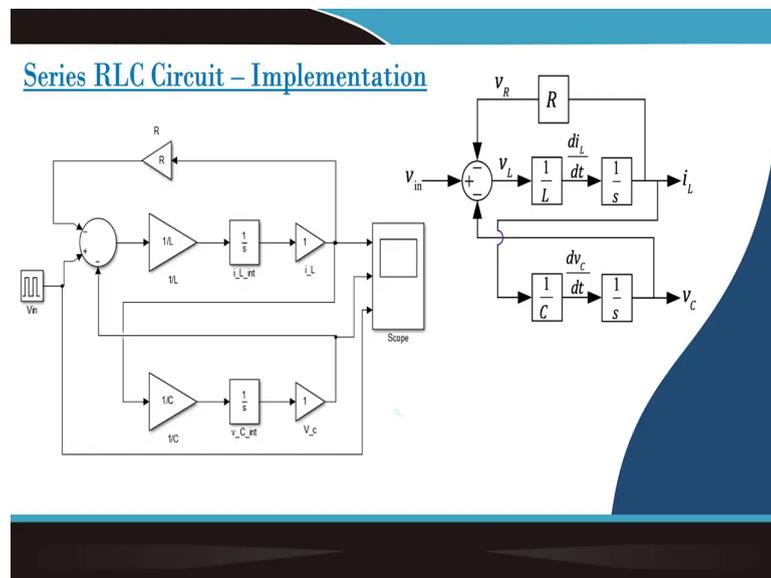
If you go to Simulink, I will show you. So, this is the block that we have already made and I have to add this particular block in the scope, ok. So, now in the scope, what are we going to show? The first signal that we want to show is the current in the inductor current. The second signal we want to show is the voltage across the capacitor. And, the third signal you want to show is the input voltage, how the input voltage is varying?

Now, we are all set. We shaped it and this is exactly what I have shown, the snapshot you know a screenshot of the Simulink block ok. Here, in order to make it more convenient, let us put you can always put a unity gain block, which is just O, ok. So, these blocks can help to just a minute. So, I can put this block here simply here. Similarly, I can put another block

here like this, block here, this is just to connect this block and I can take everything out of this, ok.

So, I can connect to this block from directly from here ok. This is for convenience. Now, we are ready to run the MATLAB and this is what I have shown here. So, this particular block is to indicate; that means, this signal I am showing i L.

(Refer Slide Time: 21:07)
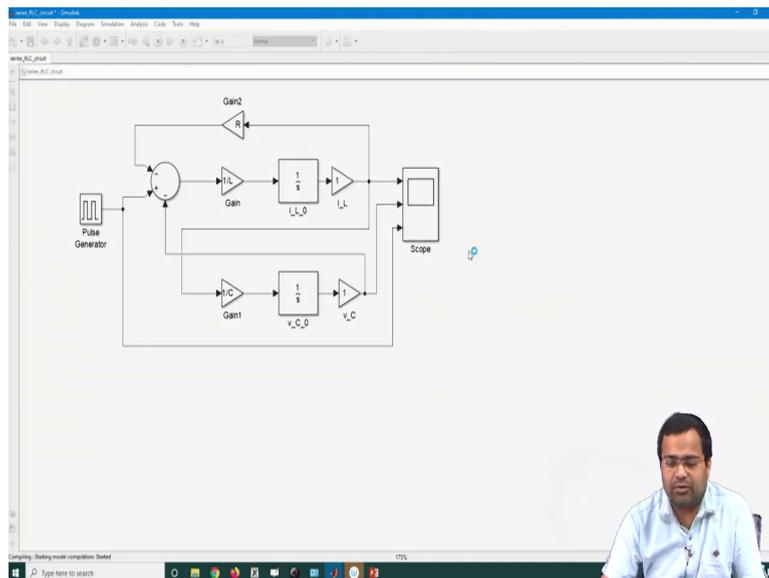
So, I can specifically write this is my inductor current and this block I am showing is the capacitor voltage in this case. Now, I am ready to run the simulation. So, it shows some error, where the initial blocks are not defined. So, I need to run this initial file initialization file.
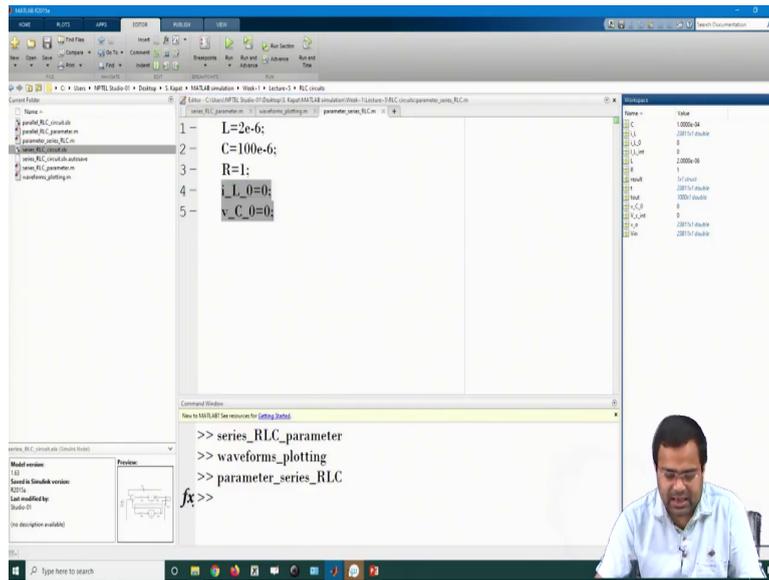
So, let us go and run this file. You need to run this file. Once you run this file, these initial values are now defined. It is placed in your workspace. Then you can start the simulation and then there will be no error.
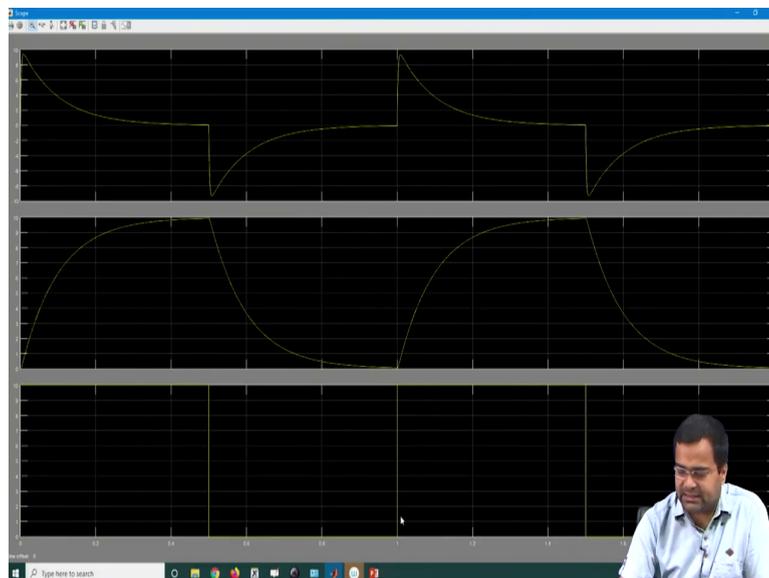
Now, you will not find any error. And it will show the; that means, initially we are applying 10 V for 0.5 ms then because you are putting 50 percent width ratio. So, it is a periodic pulse between 0 to 1 millisecond you can see and for 50 percent time it is 10 volts, then it is 0 volt,

10 volt 0 volt and so on. And, first one is the inductor current; the second one is the output voltage.

(Refer Slide Time: 22:22)
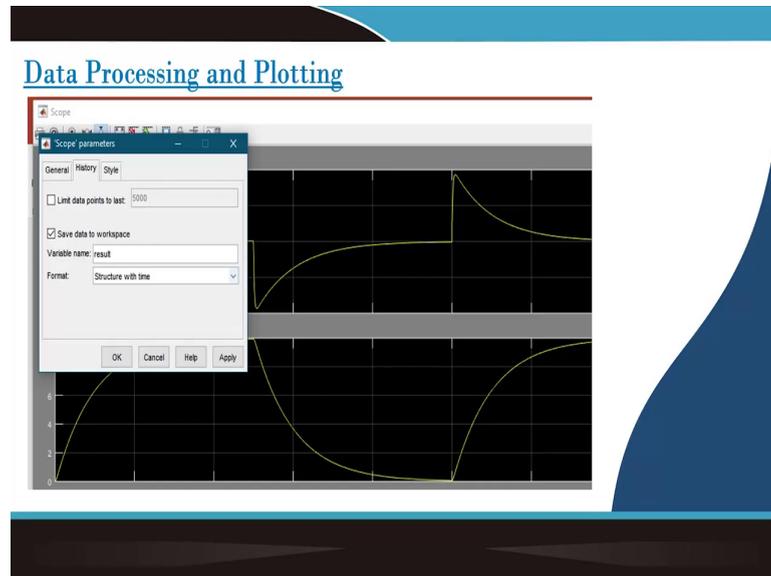
And, I have also provided a snapshot; that means, if you show the scope whatever I said the how to store the data result? And, please remember in the scope, if you go to the scope history, it should be structured with time, because I want to plot the signal. Now, I will show you the data that I have plotted actually stored here. If you go to result, this is where we put the scope data.

(Refer Slide Time: 22:56)
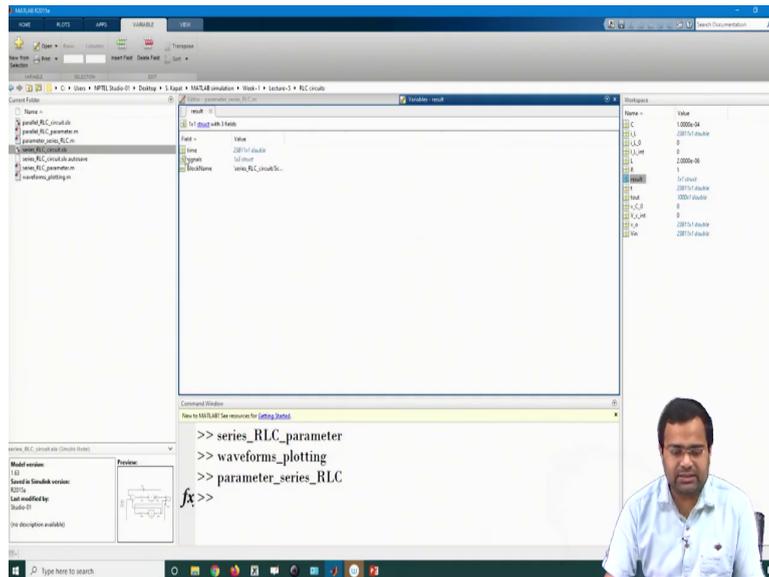

(Refer Slide Time: 23:01)
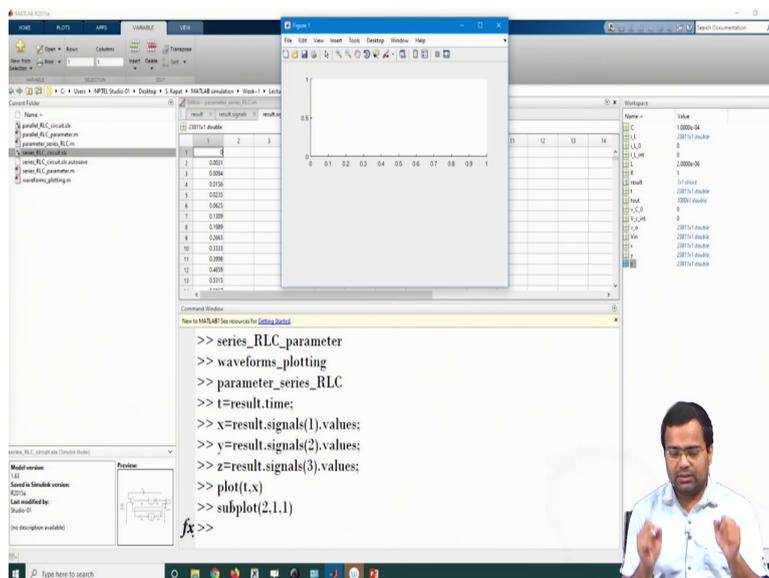
The result block, if we go to the result block, then you will see the result time is something called result dot time. You can see here result dot time, it gives you all the time series data. That means, I can write here I am creating; that means, t equal to result dot time. So, it will give me time series data. So, this time series data is now coming here.

So, it is like a 23,811 into 1, it is also 23,800 so, it's a double; that means, it's a vector which has how many elements? 23,811 number of data are there, which are in a placed in a column

matrix, 1 column like a 2000 23,811 number of rows and 1 column. Now, I am interested to see, what are the axis of data? So, these are the signal.

And, since signals means the scope input, because scope there are three inputs are there in the scope. If you see the scopes have three input, but scope in order to display it input we need to plot the time data; that means, at different time instant, what are the values of different signals? So, the time vector we have already stored.

(Refer Slide Time: 24:36)
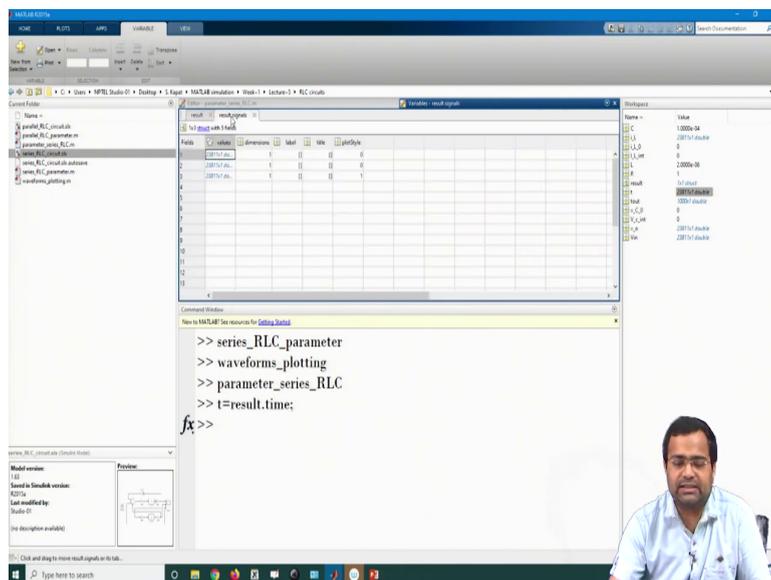
Now, in the signal if we go to these signals; that means, result dot signals. It has three different channel; that means, 1, 2, 3, these are the three inputs. The first input is this, second input is this, and third input is this. And, if we click on this first input, then it will show result dot signals this value. So, we can store like x equal to result dot signals 1 dot values, it stores the x axis data, you can see x axis data that are stored.

I can do it on the y axis; that means, Channel 2 I can create a variable like y, but interestingly you will see y and I can also create another data for z 3, that is in the third input that is the z. So, now we got all the data's here and we are ready to plot. So, we have time series data vector and you see the number of elements in t vector are same as x, y, z, because the number of time instant and their corresponding value, they must be consistent.

(Refer Slide Time: 26:13)
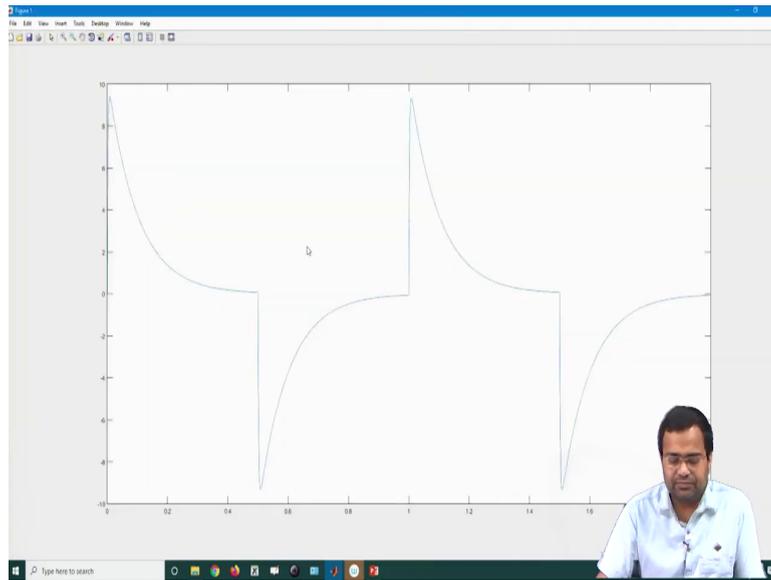
Now, plot t comma x, it will show my time domain waveform of inductor current. And, in this particular case, now I want to see the voltage and current together; that means, you know instead of creating a separate figure; I want to show sub figure.
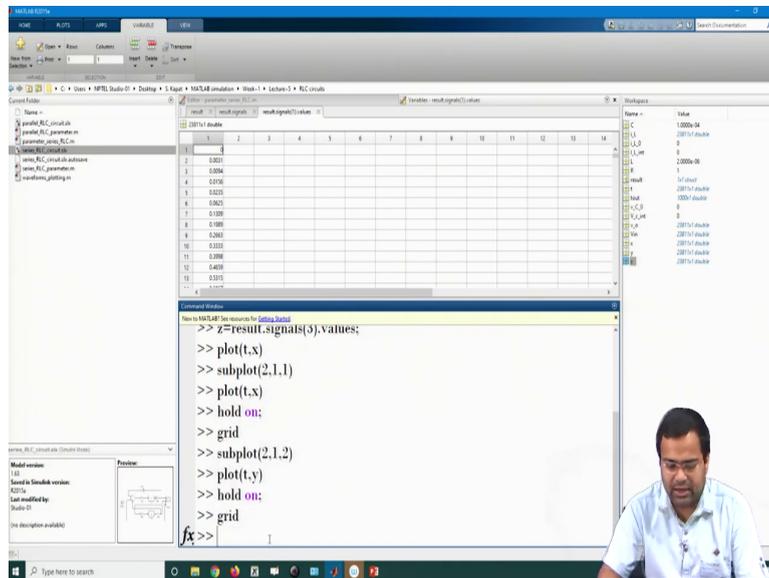
(Refer Slide Time: 26:56)
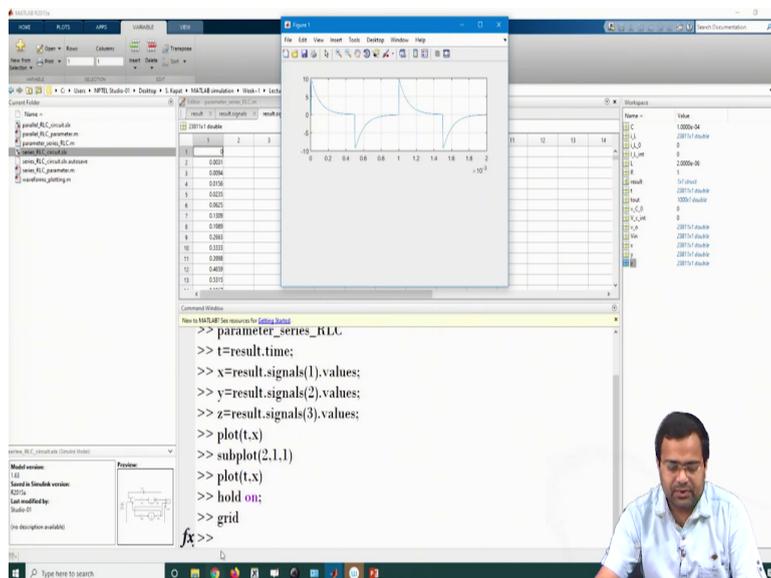
So, you can show sub figure a subplot, I want to create a subplot of 2 comma, 1 comma 1; that means, it will show in terms of 2 row. First row it will show the plot which we want and

in this case we want to show the plot of t 1 axis, I need to hold on and we can also grid it; that means, it will show the grid. So, it will show grid.

(Refer Slide Time: 27:10)

Next we can repeat the same we can do subplot 2; that means, the second one, if you go to the second one, then we can repeat it. Now, we are plotting the voltage and we can again hold on; that means, if you want to put another plot, it is also possible, by that way we got the second plot.
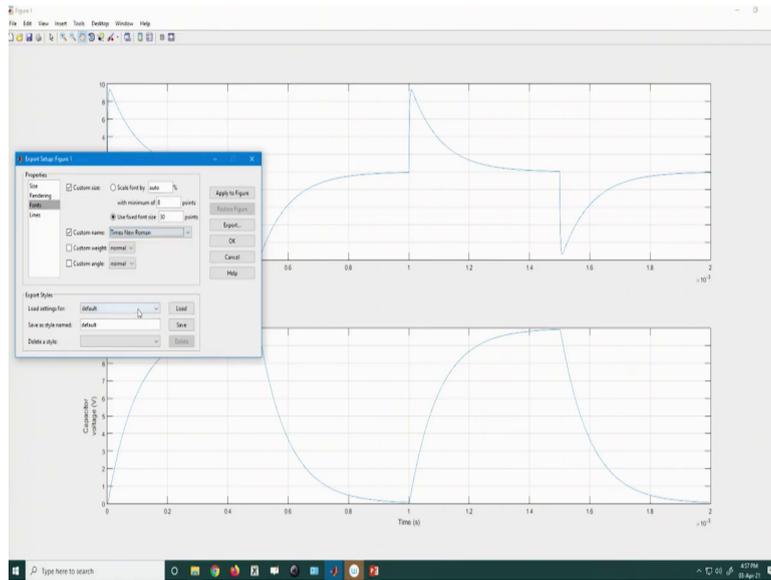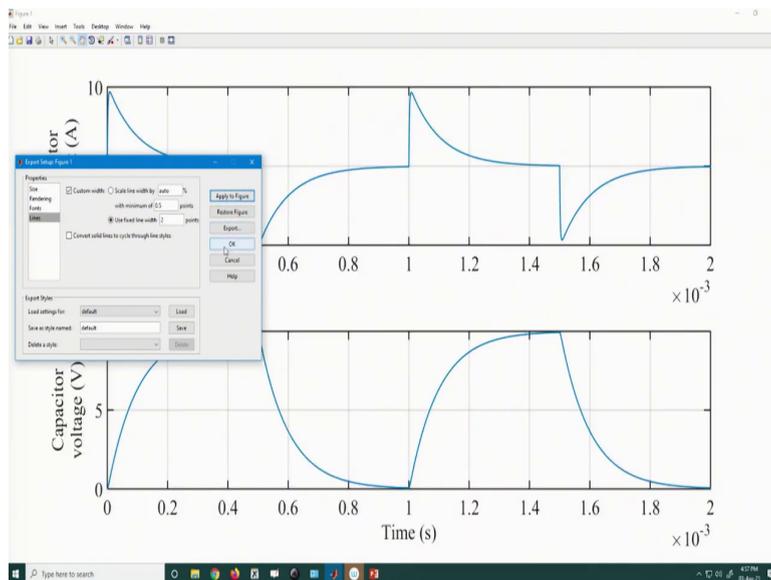
(Refer Slide Time: 27:29)
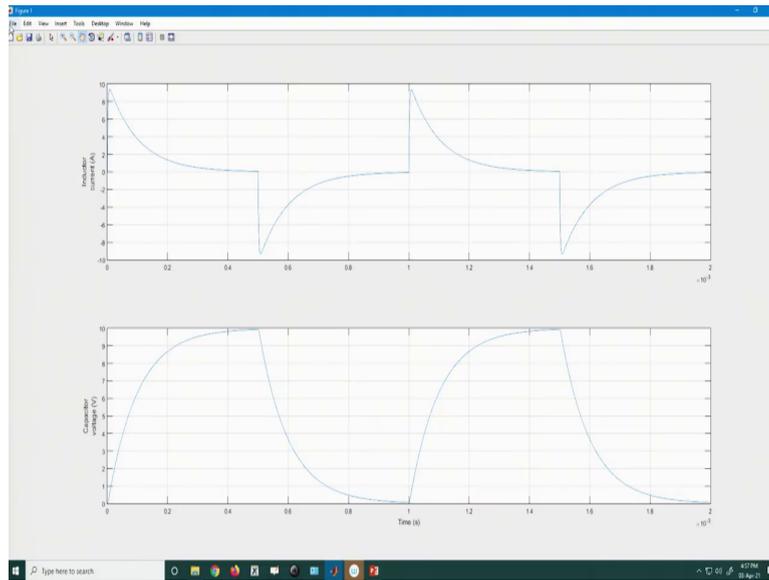
So, the first one is the current and the second one is the voltage. Now, after we complete the plot commands, it is not clearly visible. So, we can put x level as the time, it is in second, because in this it is 10 to the power minus 3 is there. Then, if we select that particular channel; that means, this plot then we can insert the y level, which is my output or the capacitor voltage capacitor voltage of voltage it is in v.

And, for this plot we can also plot y, which is inductor current. So, now, we want to go to the export set up.
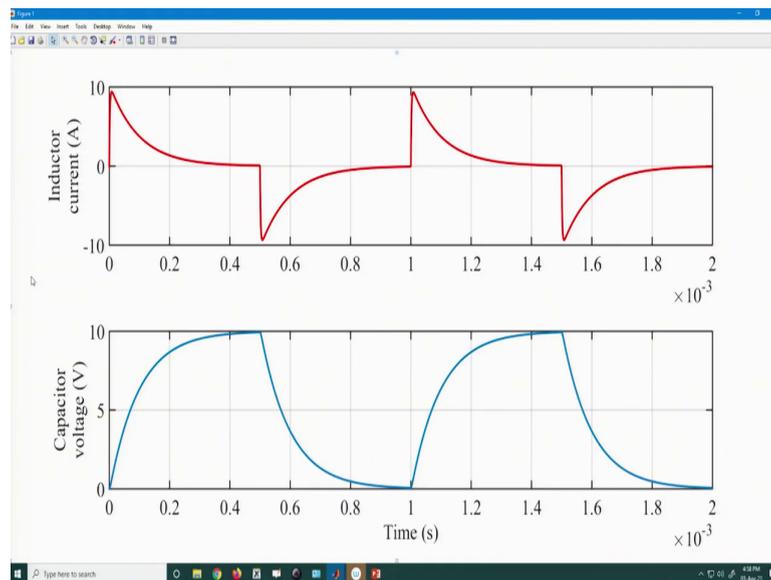
(Refer Slide Time: 28:38)

(Refer Slide Time: 28:55)

(Refer Slide Time: 29:00)

Where we can increase the font size, I want to increase to 30 and it is better to use the Times New Roman. In fact, times will be enough. Then, line width I want to increase to 2 then apply. If we apply, you know now the width has increased. Now, we want to if we want to further increase. We can select the right click. We can change color if you want to let us say red color wave form. In fact, if you want to increase the line width, line style, everything is possible.

So, line width, line color, anything that we want we can increase it ok. So, by that way we can actually change the, this particular plot. So, these are the plot command that is used for simulation of, times series data; that means, what we learn. Now if we go back, this is my series RLC circuit.

Now, I can develop a parallel RLC circuit. Now, since we already know how to develop a model. So, we will directly jump go into parallel RLC circuit. What is there in the parallel RLC circuit? That means, let me go back to the Simulink file.
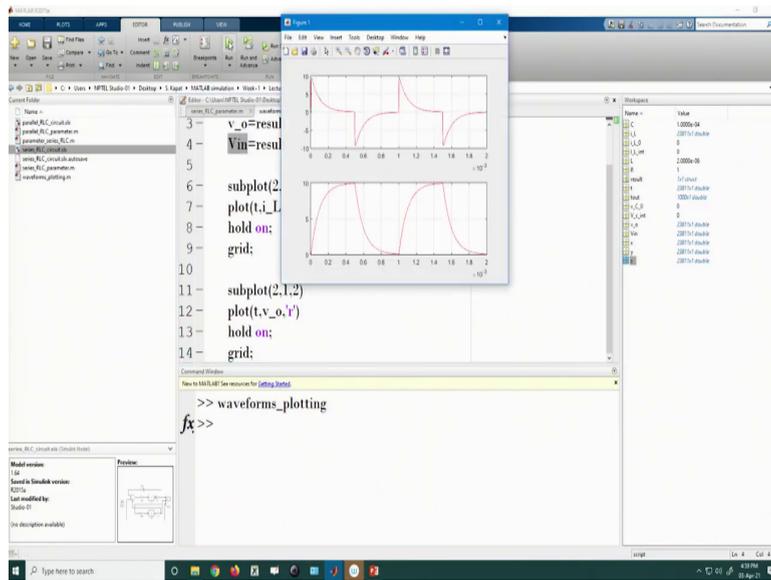
(Refer Slide Time: 30:12)
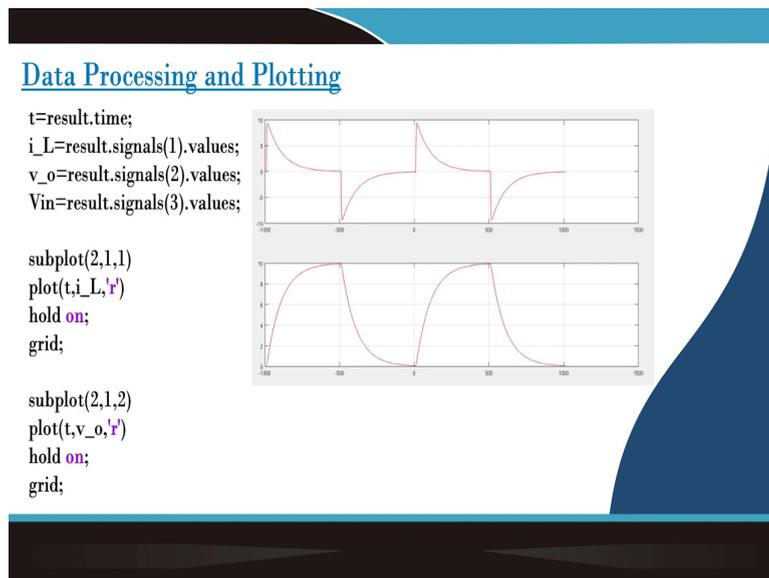
So, series block we have implemented. This is the plot I am showing. So, I have given some plot command. So, in order to make life simple, you can create a separate file; that means, you know whatever I have done here; I have created a separate file.

So, first one is the inductor current, the second one is the output voltage, and the third one is the input voltage and you can do subplot then figure all this, you can if you just simply click it will be enough, it will plot.

(Refer Slide Time: 30:54)

So, this separate file is used for simulation purpose and which I have shown in this particular data processing part.

(Refer Slide Time: 31:06)

The next part, I want we want to implement parallel RLC circuit and this part using this Simulink block and once we develop. So, before we go to the parallel RLC circuit, we need

to create a subsystem, because suppose we want to connect multiple such series circuit in parallel or in series so we can create a subsystem.

(Refer Slide Time: 31:37)

(Refer Slide Time: 31:43)

So, how do you create a subsystem? That means, if we select that particular block and if you right click, create subsystem. And, if you go inside the subsystem, all our blocks will be

there. It will be retained. Here we will name it. The first block will name it as v capacitor. That is my capacitor voltage. This block already exists.

So, I will name like a v C ok, this block let us say some v and this block i because there will be no confusion. So, this is the output port i L and this block is my input V in.

(Refer Slide Time: 32:29)
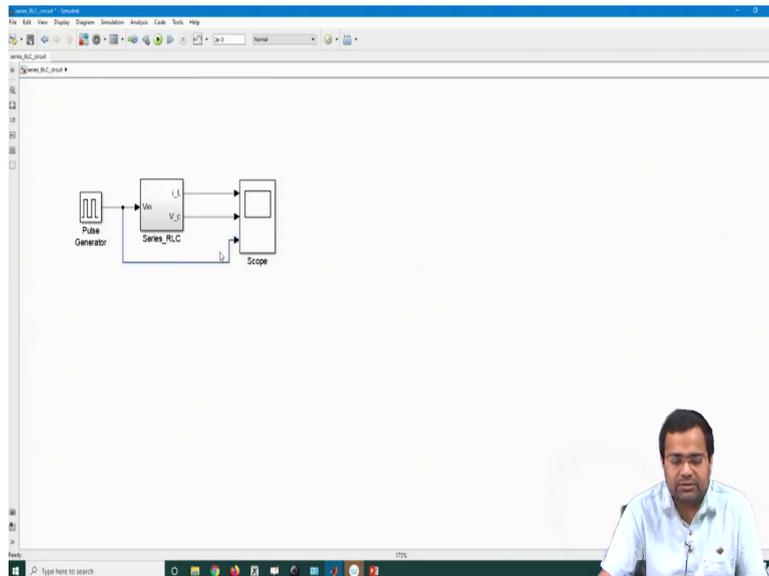
Once we go back now, it will show my subsystem and you can name it as a series RLC. So, this is a series RLC circuit. Now, if you want to create, if you want to place another series RLC circuit in series or parallel, it is possible. I can show you in the next block, but this block is now ready for you.

So, now you want to show the next block which is the parallel RLC circuit ok. So, that means, subsystem creation is simple. If we create subsystem it will make you know my simulation easy and is more portable, because I want to you know do it for different blocks and so on.

So, in this particular presentation, my aim was to just introduce the MATLAB. And, we have implemented a series of algebra, which is very simple, but in the next presentation I want to build up another model of the converter. So, that it will help you know whatever we do, whatever we learn, it will help to do a MATLAB simulation custom building block.

So, thank you very much, I think we will go to the next presentation.