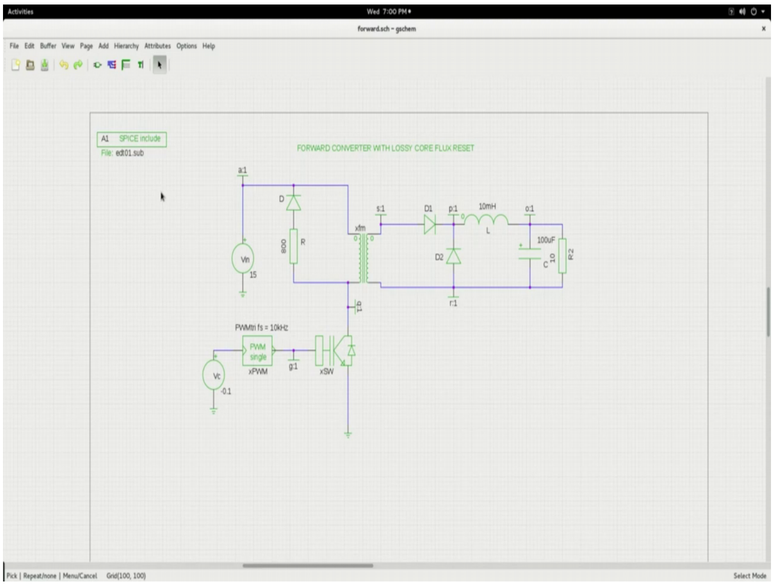
Fundamentals of Power Electronics
Prof. L. Umanand
Department of Electronics Systems Engineering
Indian Institute of Science, Bengaluru

Lecture - 56
Simulating with lossy core reset

Let us now see how we go about simulating the forward converter circuit. Forward converter with lossy core flux reset calling it lossy, because this is the core reset circuit which we studied and this is the lossy resistor which is going to dissipate the magnetizing energy out of the core.

(Refer Slide Time: 00:35)



So, this is essentially the forward converter circuit that we have been discussing till now. This is the power semiconductor switch which is going to turn on and off and it is getting the gate drive signal from PWM, block and the PWM block is given with a control reference of minus 0.1.

Remember last time when we were discussing the PWM block we, we said that this triangle within the PWM block is transiting from a minus 1 to plus 1. So, at 0 if the control signal is 0, you will get 50 percent duty cycle. If the control signal is minus 1 then you will get 0 percent duty cycle, if the control signal is plus 1 then it is 100 percent duty cycle. So, at minus 0.1, it will be around 40 percent duty cycle.

So, this is, the gate drive signal which is given to the BJT switch. Now, if we come to the transformer portion the transformer is not the regular transformer provided by spice that is a linear transformer model, we have included a sub circuit model here so, that even saturation effects can be taken care of. One could also use the core based model of spice here, but this is a generic model, any good model of the transformer can be used without loss of generality. Now, this portion of the circuit as you already know is the core reset freewheeling circuit, we have a resistor and the time and this is of course, the input and we are applying 15 volts DC here.

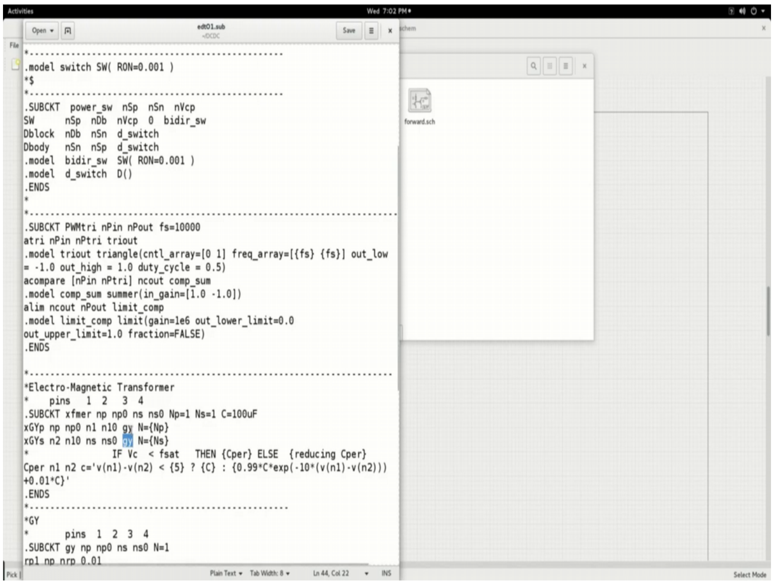
On the secondary side, we have the diode D 1 and D 2 and LC and you can recognize the buck converter topology in the output side. Now after that for the display of the results I have been telling you to label the nodes like this. So, you have the node A and this is node Q, if you want to see the voltage across the device here there is a node S, here for at the dot end of the transformer. Secondary there is a node P indicating the pole voltage of the buck converter and the node O; you should remember that the output side is not at the ground potential, this is not at the ground potential. So, you should because it is isolated from the primary side. So, whenever you want to measure a signal here, it should be with respect to some point on the output side. So, let us say we have a reference node R here. So, will causes the reference node and anything on the secondary side, we will measure it with respect to this reference node.

So, when you want to see the secondary voltage, you will say VS with respect to R VP with respect to R that is V P comma R VO comma R then the voltages are with respect to this reference node. So, I have put in some typical values here.

In fact, these are the same values which we used for the buck converter. And you can use the similar thing for the forward converter example here too. Using the PWM with a switching frequency of 10 kilohertz and this is the spice include which we have been doing for all the simulations and I have included the EDT 0 1 dot sub file.

So, this file, let us simulate. Now going back to the folder the forward dot CIR the forward dot CIR contains dot TRAN attention analysis statement 0.1 micro second step and up to 10 milliseconds use initial conditions.

(Refer Slide Time: 05:09)

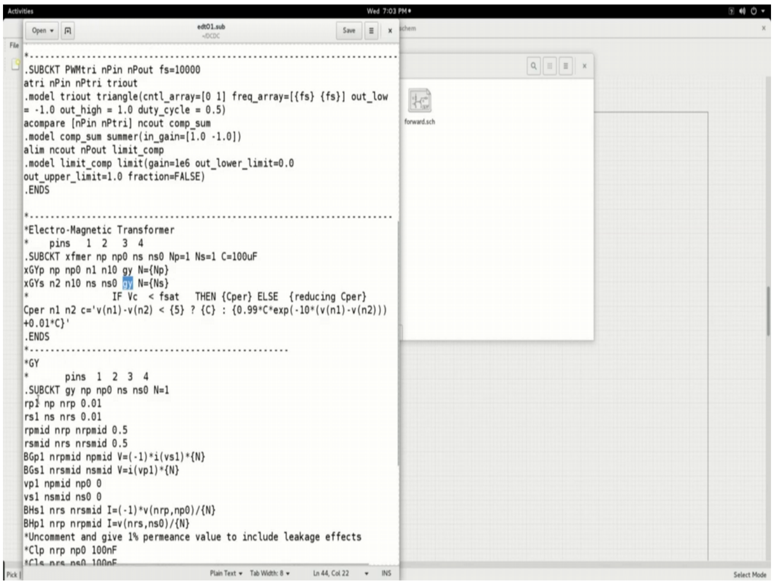


Then dot include the forward dot net forward dot net has to be generated which we will do now shortly. Now coming to the EDT, let me close this. Now the RDT dot sub I have already included the model for the electromagnetic transformer. So, there is a model for the electromagnetic transformer, the primary dot point primary non dot point secondary dot point, secondary non dot point there is a default primary number of TRAN secondary number of turns and C, here C is not to be thought of as capacitance , but as permeance.

So, within the magnetic domain the permeance in the core behaves very much like the capacitance or capacitor. So, in spice the equivalent component would be A C. And therefore, we have used C, I will explain more about this capacitance permeance relationship. Later on when we are discussing the magnetics so, we have used this kind of the model here. We have also put some kind of saturation effect here, if the value the permeance if the value of the flux is greater than a particular value then we will not, we will take the same value of C as is given here, but if it is greater than the value a particular set value, then it will start exponentially decaying down to 0.

So, this will emulate the BH curve of a core. So, this is a saturating type of a magnetic circuit and therefore, this transform a transformer has is a generic transformer which behaves much more closer to reality. Now this transformer uses to further models called gyrators and the gyrator is modeled here.

(Refer Slide Time: 07:35)

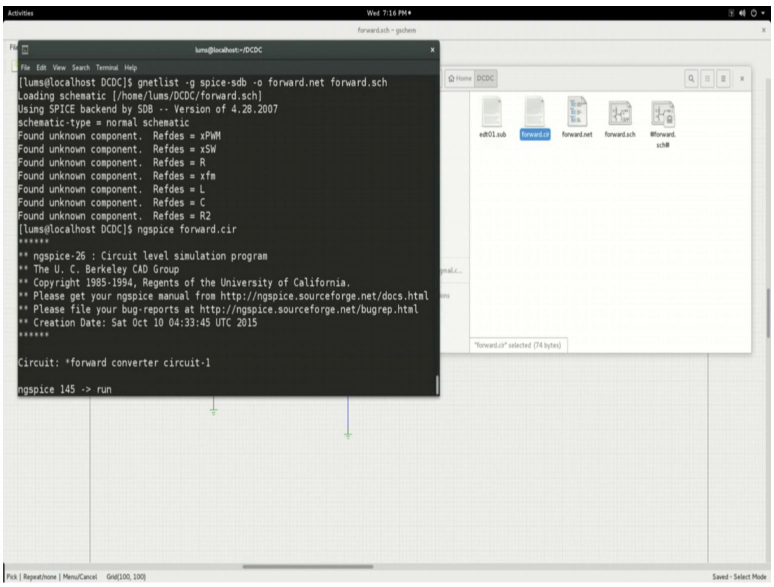


Anyway this is the model of electromagnetic transformer and you will find many models of transformersin the internet, you can use anything that is suitable to you.

And you can also probably use the spice provided transformer and even the linear transformer to at least get the concepts, only if you want to look at saturation then you will have to include something to make the core saturate, ok.

So, we have the EDT 0 1 sub dot sub file, also in place now we have to go forward and do the simulation using the ngspice engine which is what we will be doing.

(Refer Slide Time: 08:25)

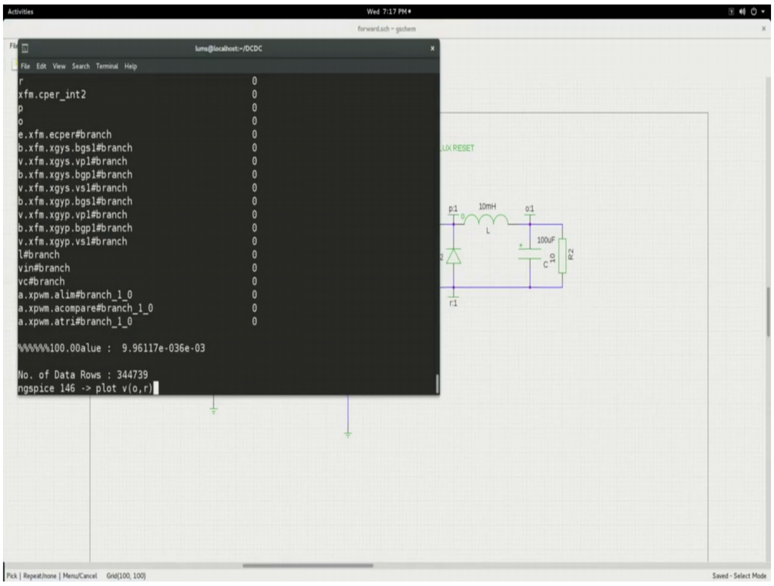


Now, so, let me go to the terminal and inside the terminal I will go to the ok. So, we are in the directory and we will generate the netlist and also call ngspice after we have generated the netlist, let us now generate the netlist and go forward.

So, generating the netlist is with this. Now familiar command G netlist dash G spice dash S DB dash O forward dot net has to be generated from forward dot SCH of the schematic file. So, from the schematic file, you give this command and now you would see that there is a net file generated.

Now we can use this net file this net file is being called in the forward dot CIR and ngspice can perform the simulation. Now all the ngspice engine, ngspice forward dot CIR. Now this will load the circuit schematic with the netlist and now run to perform the simulation.

(Refer Slide Time: 09:39)

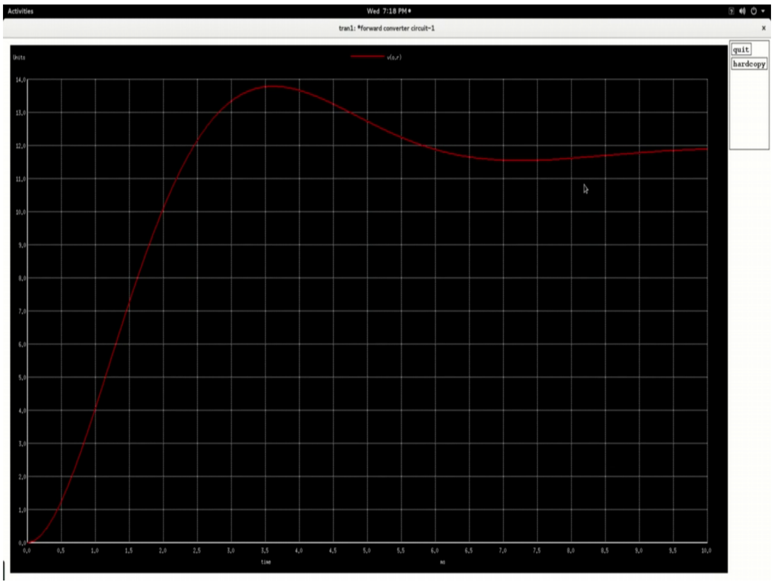


So, this will take some time I will later on tell you methods in which you can reduce the simulation time. The first time simulation will take quite some time basically, because you would have given an extended range till it reaches steady state. After it reaches steady state, you can then note down the steady state value and give it as initial conditions. And I do the simulation with those initial conditions then you will see that it in just a few cycles, you will quickly get the wave shapes and wave forms that you actually would like to see at various parts of the circuit.

So, now, here you have some time for 80 percent of the simulation done, you just have few more moments to wait and you have the complete simulation done now. So, you could plot. So, when you plot let say I want to plot out do not do not just put V in brackets O.

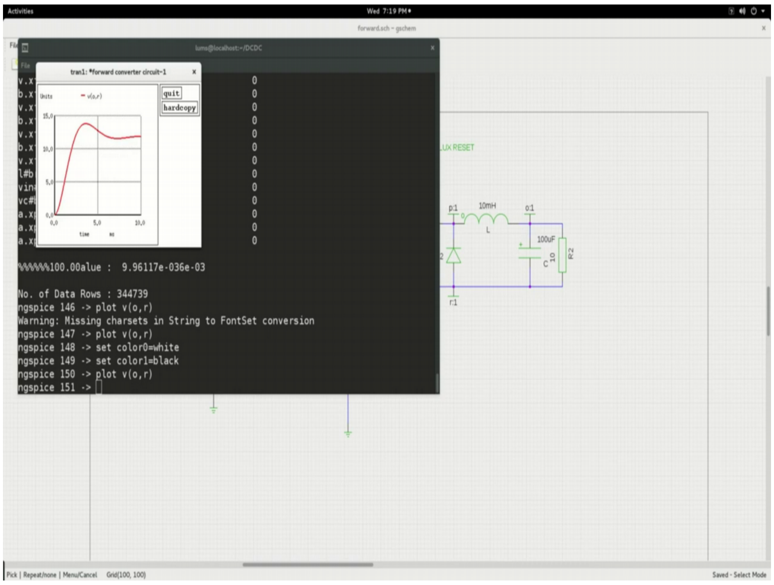
Now with respect to R with respect to the R node, because observe that we want to see the output with respect to this R node. So, it will be V O comma R. So, coming back here, you will see that this is the output ratio.

(Refer Slide Time: 11:05)



Now let us, before we further investigate few more aspects of the waveforms and waveforms of the other parts of the circuit one tip here would be, ok. Let me put that back here, you have a black background and on the black background, you have the white grid and the waveforms showing up in color here. In some cases, you would prefer to have a white background and have black grid lines and then the waveforms in color. This is especially useful when you want to document it or cut I will take a screenshot of the plot out result and then put it into your document

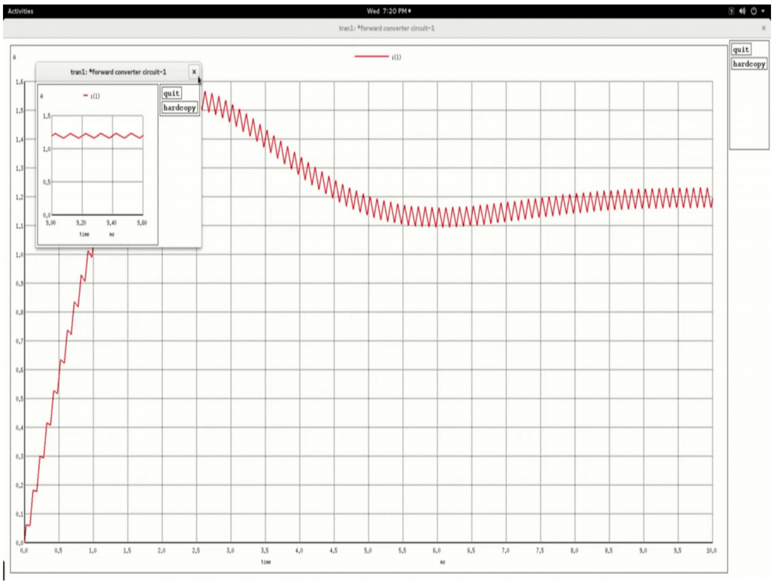
(Refer Slide Time: 12:03)



So, to change the background what you could do? You could use this following commands. You can always go back to the ngspice manual and check out these commands color 0. So, color 0, I will now set it to white. So, this basically sets the background color 0 represents background color set, these are fix it reserved words color one is the foreground which we use set it to black.

So, now, same plot command would appear in much deserved black a white background with a black foreground, ok. Now with this let us see few more waveforms of interest to us. So, we would like to see the, inductor current waveform here. So, let us have a look at the inductor current waveform plot i l.

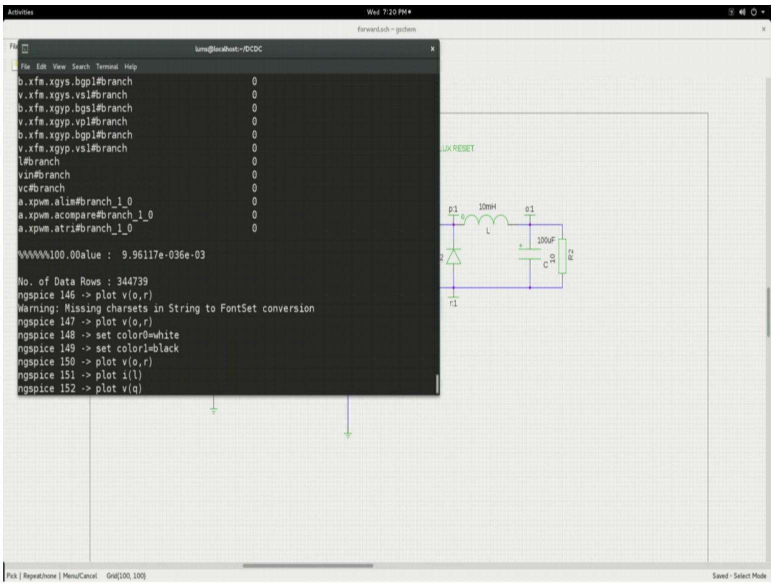
(Refer Slide Time: 13:11)



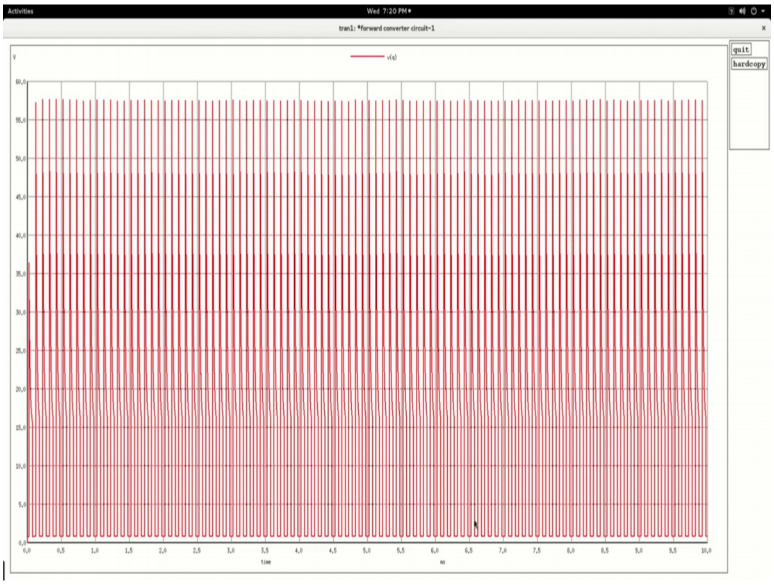
And see as expected the inductor current triangular wave shape; this is what we are expecting. And you observe that later on around here after the 9.5 millisecond almost it is trying to reach stable point, you could probably give the initial condition at around this value. So, that quickly, you will see your circuit coming to a stable state from the initial condition that is a very nice approach do to shorten the simulation time, if you are not interested in the transients.

Now another the point that we would like to see is Vq here, we have been discussing.

(Refer Slide Time: 13:59)

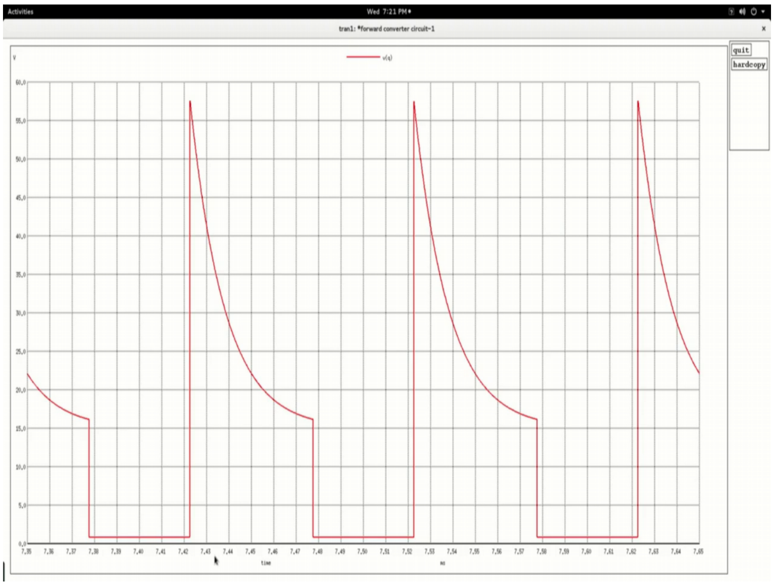


(Refer Slide Time: 14:07)



So, much about v q, you will take across the transistor. So, let us say v q, v q let us expand it to just a few cycles and you would see.

(Refer Slide Time: 14:19)



Now, you see this is exactly what we have been discussing this is the on time. So, therefore, the on state then the moment the BJT switches off. So, you see a high spike and then the current decays exponentially and goes towards vvn value or vcc value.

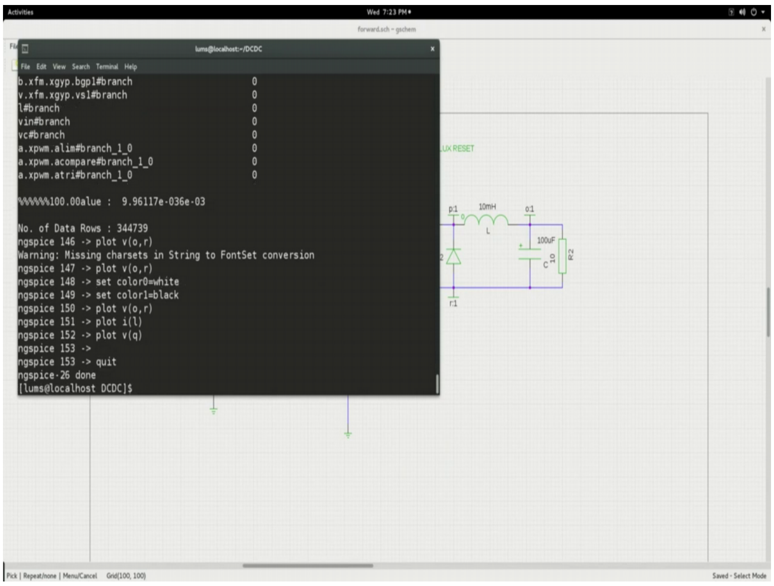
So, this is typical of the lossy flux reset and this exponential decay is the LM by R time constant and this is due to the decay in the magnetizing current. So, this is one important waveform which I wanted to show you., other another point important point of waveform that we would like to see is the currents that are flowing through this. But unfortunately here there are no see if it is an inductor, inductor being a state energy storing variable component the current is a state variable and then it will so, give the branch current of that.

If it is a resistor or any other branch, you will not see automatically with the branch currents available here. You, you observe to the branch currents are available in the list of plotouts only for L branch. If there is a source vin and vc branch. So, you see that there will be currents available only for this vc a source branch another source branch and NL branch. So, how to see and measure currents at various other points?

So, I will give you one more trip, one more tip here we could put 0 sources wherever we want to measure the currents and then on simulation, you will get those branch currents and then you could see those branch currents.

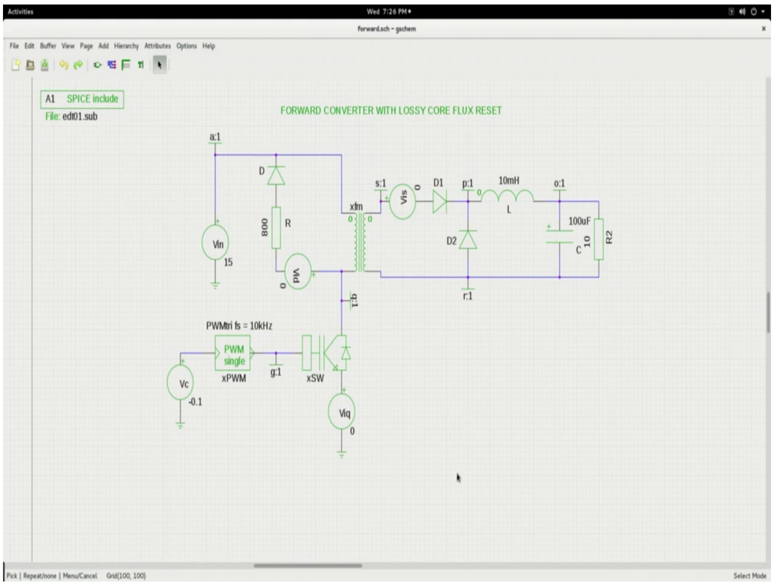
Let us see we can do such a thing and the meantime first what I will do is quit this plot quit and clear the screen.

(Refer Slide Time: 16:31)



And now, we will do the modification here. So, what I am going to do? I am going to source like this I am going to introduce here, here maybe here. And here no what we could do, we need not have two here, we will have one here and one here. So, these two will give you all the information of the current which flows here and one here which will give you the secondary current. So, three voltage sources with 0 voltage value we will interpose inside in this circuit schematic.

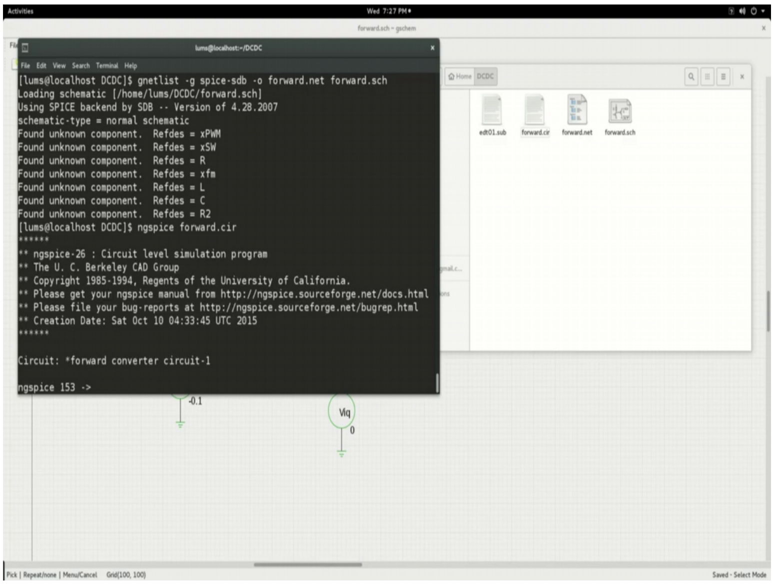
(Refer Slide Time: 17:17)



So, now you see, I have made the changes which I said I would do which is to put a source here a source here and the source here. Now observe also that I have put 0 voltage value at all these source. So, these are just now behaving like current sensors, now I have put the positive at in such a way that the current flows into the positive. So, here we are expecting the current to flow in this fashion. So, it flows into the positive, we are expecting the current to flow in this direction here also it flows into the positive and likewise from the dot end into the positive. So, if you put the source in such a way that the positive is in such a direction that the current flows into it then you will get the proper direction of the current ok.

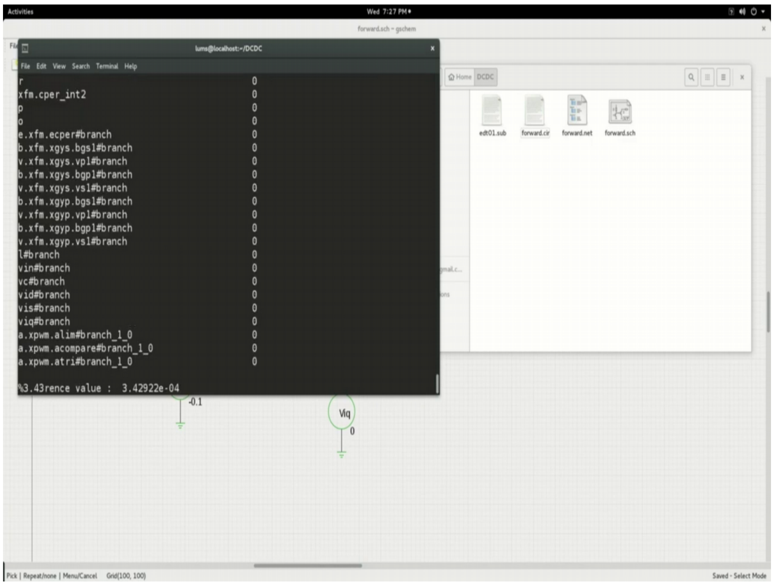
So, now, this otherwise there is no other change that we have done, we have output current sensors. And with this current sensors, we will be able to get more insight into the circuit. So, let me see this and do the simulation once again.

(Refer Slide Time: 18:31)



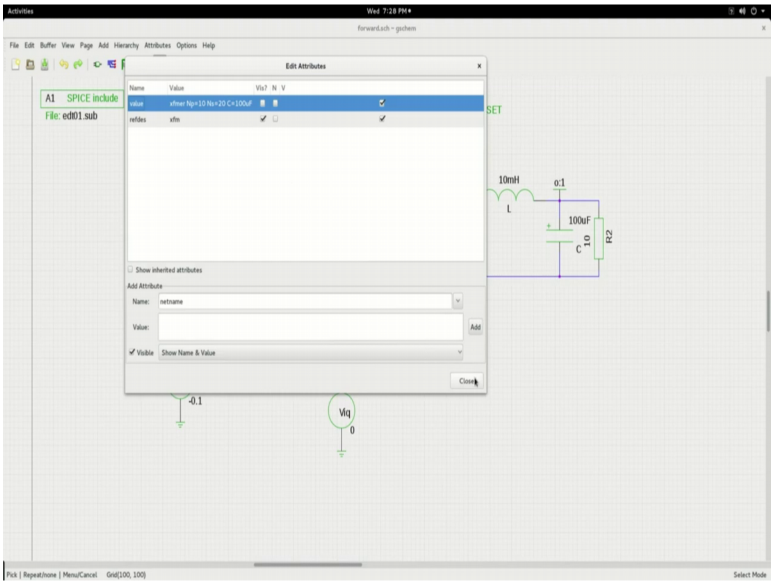
So, let us generate the netlist, netlist is generated. And now we will go into NG spay of spice and run the simulation once again.

(Refer Slide Time: 18:47)



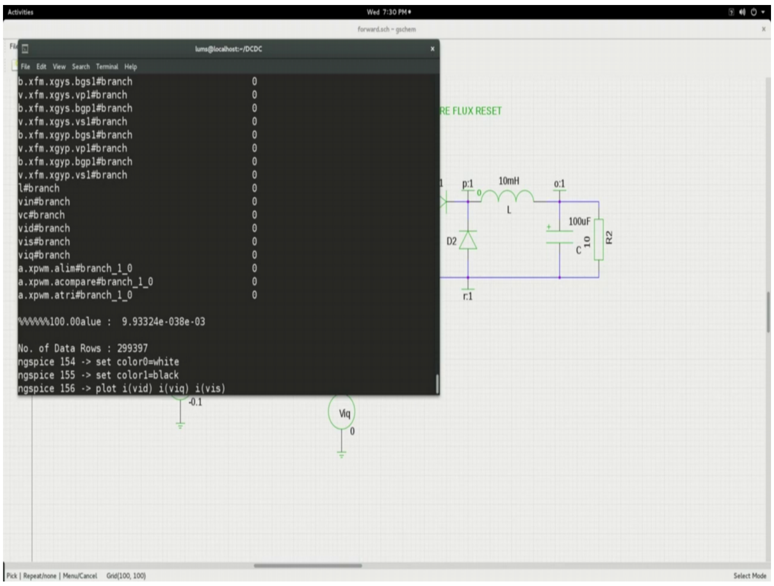
So, it should take some time then after that we could we could change the background color to white and the foreground color to black. So, this would make a much more presentable display and a display which you can later on use it for documentation purposes. Now, while the simulation is going I would like to also show that in the transformer, when I double click on the transformer.

(Refer Slide Time: 19:29)



It is sub circuit we, I have put here the primary number of turns 10 secondary number of turns 20 and the permeance is 100 micro. So, this is the parameters that is passed on to the transformer. So, therefore, you see that NP 10 and NS 20 means there is a trans ratio of 1 is to 2.

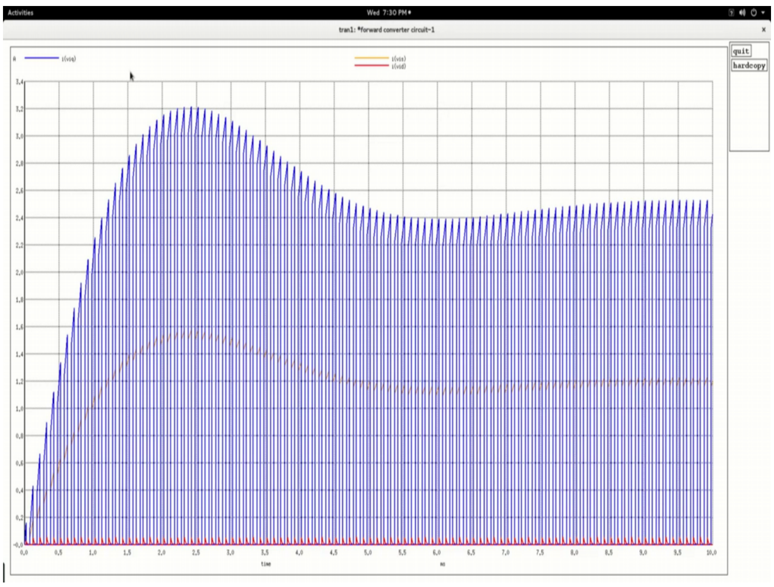
(Refer Slide Time: 20:03)



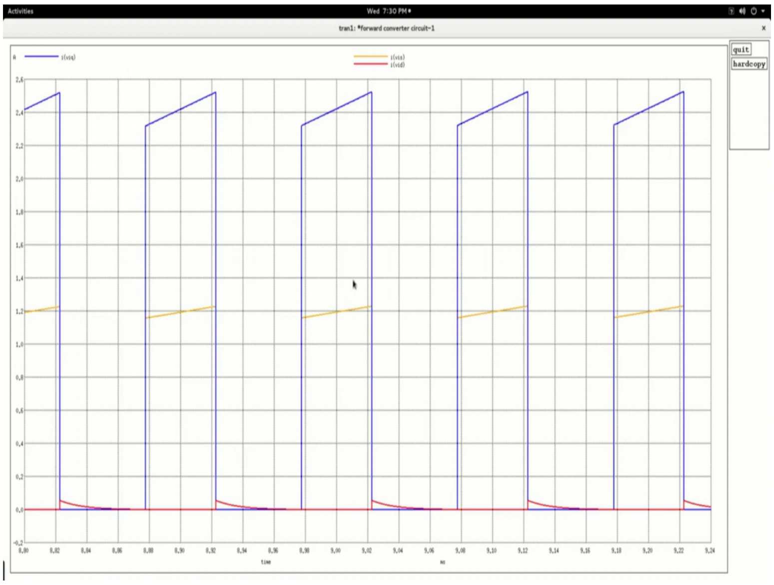
Now coming back to here the simulation is over. And let me set color background equals white set color foreground equals black. Now let me plot, now what a plot. Now I would like to see the switch current here. Now the switch current remember is nothing, but the reflected part here of the secondary reflected part and there will also be a magnetizing part. So, these two components together form the switch current.

And during the time of the switch is off the reflected part is not there, there is only the magnetizing part which will decay. Now this effect, we will be able to see clearly. So, I will view this current this current and this current. So, I of vid i of viq and i of vis. So, let us i of you could also see here you have the vid branch current vis branch current, viq branch current. So, voltage of the demagnetizing winding diode winding vid i of the current flowing through the switch viq and the current flowing through the second wave vis. So, let us plot that let me expand this and you will see that and let me also probably see just few.

(Refer Slide Time: 21:43)



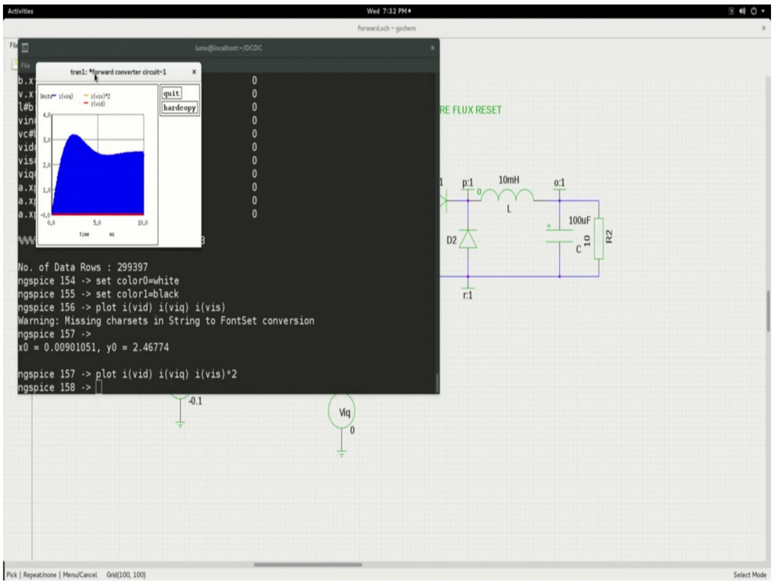
(Refer Slide Time: 21:53)



Now, you see here, viq blue viq, this is the switch current. So, the switch current is like this, it is the blue portion clipped. Now this is actually N times I naught recall that this point here where the cursor is would be N times I naught. Now this is actually the I naught current. So, if I plot the inductor current, it will flow all along through like this. So, therefore, it may be a good idea to plot that also I will do that later. So, this is actually the secondary current and you see the red one is the D magnetizing current see the it is so, small in magnitude. So, actually the magnetizing part will take a linear wave shape here and goes in an exponential manner and from here again linear and so on.

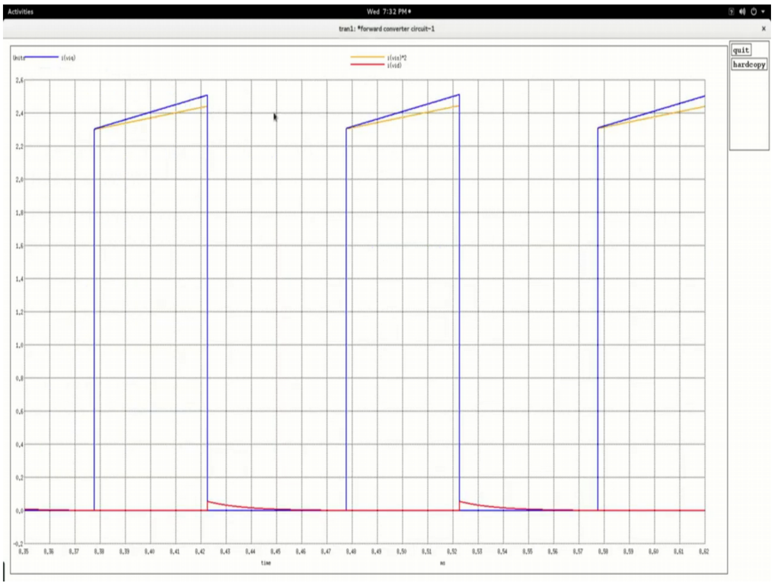
So, let us say if I say N times I naught, this will is supposed to match here exactly except for the magnetizing. So, you will see that this will more or less come in this like this here like a small triangular gap would be left here that would be the magnetizing part.

(Refer Slide Time: 23:37)



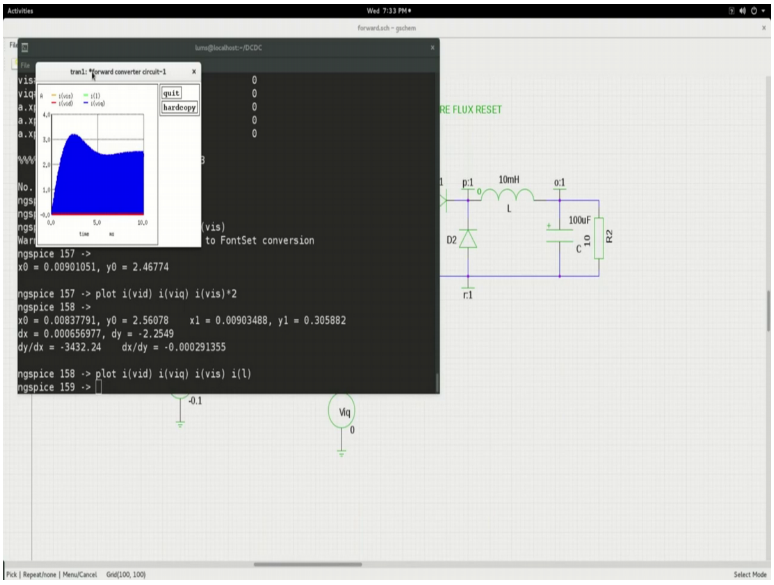
So, let us see that what I will do is now I will give the this is actually the secondary current into N and N is to in this case. And now plot that and let me expand that.

(Refer Slide Time: 24:01)



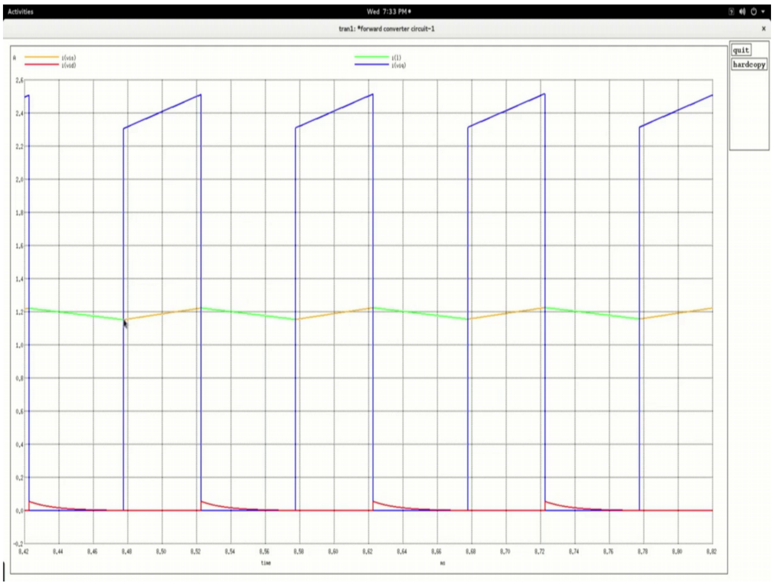
Now, you observe now you see this, the primary or the secondary exactly same except for the difference which is the magnetizing part. So, the primary switch is composed of the reflected part and the magnetizing part and the magnetizing part actually is an amount equivalent to this which will come and match exactly here and then this will start exponentially decaying.

(Refer Slide Time: 24:45)



So, that is one aspect I would also like to compare with the inductance, inductance waveform inductors waveform. So, let us say we also have I will remove this and also put i of l the inductor waveform the interesting to see that let me zoom in.

(Refer Slide Time: 25:01)

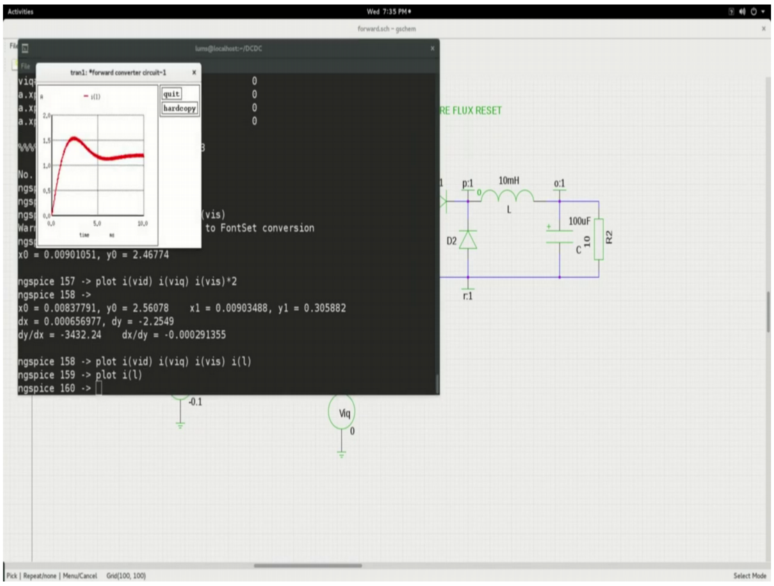


Now you see the green is the inductor waveform, you see that this inductor waveform the part when the switch is on is the same which we are measuring as the secondary current. And that is actually what is reflected to the primary. Now the central the average of the inductor waveform is I naught and that will come as the flattop equivalent of the primary switch current which is N times I naught. So, the primary switch currents flattop equivalent equivalent flattop current will be NI naught in this case.

Now I would like to show you one more thing before closing the simulation and that is if you want to do the immediately some changes and check the waveform that various points, you would not like to wait long. And then wait for all the simulation to run through and then wait to see what is happening here. First off one thing what you could do is you could take the value of the state variables. If you see there are two state variables the inputs and the state variables define the system and inputs are DC in this case and the state variables are il and vc.

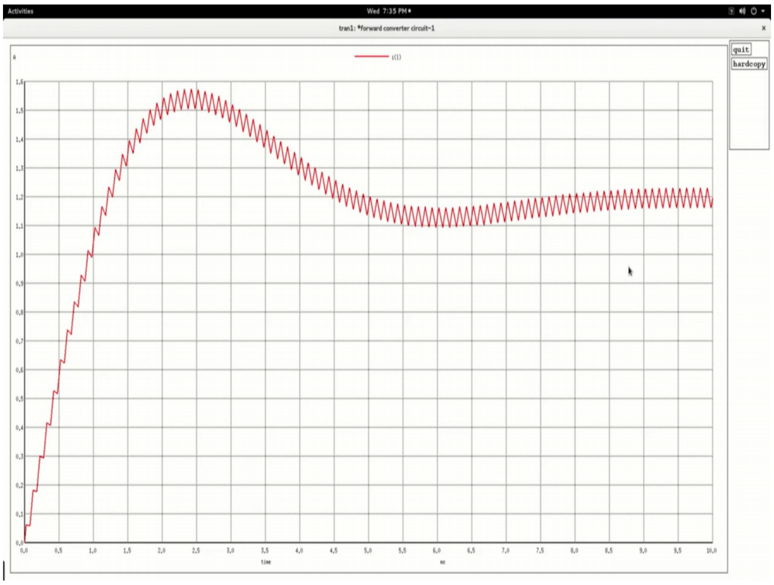
So, let us take the value of il and vc at the end of the simulation plug it in as initial conditions here and then you will see the simulation can be much faster. So, that is one exercise I will just show you.

(Refer Slide Time: 26:59)

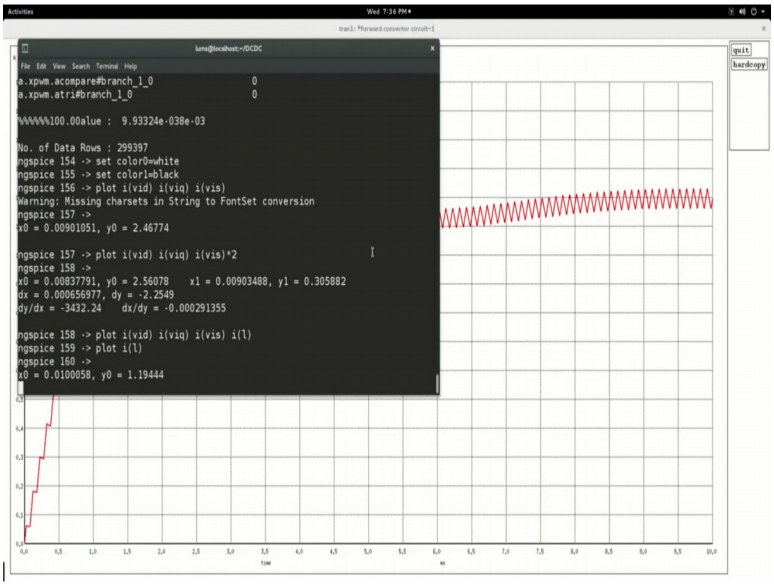


I will just quit from here oh no not quit, I will just plot plot first IL 1 state variable.

(Refer Slide Time: 27:01)

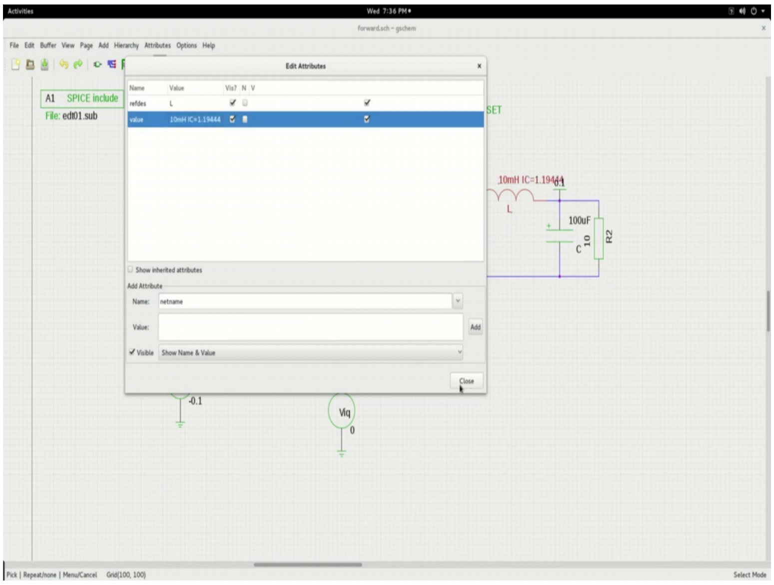


(Refer Slide Time: 27:09)



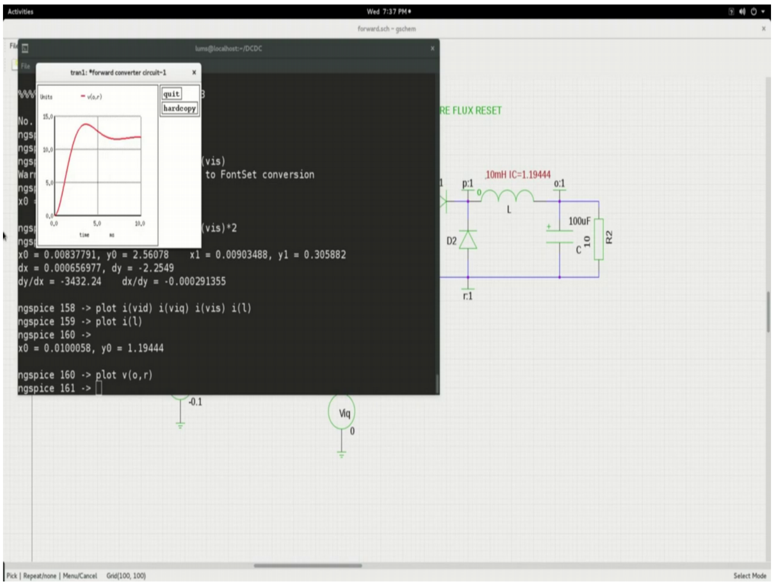
So, let us take the value at the end of the simulation here click on that and then you will see the values one point had at that final value, it is around 1.19444 so, that I will use it as the initial condition for the inductance. So, when I click on the value for the inductance millinery, I will also put initial condition equals 1.19444 .

(Refer Slide Time: 27:31)



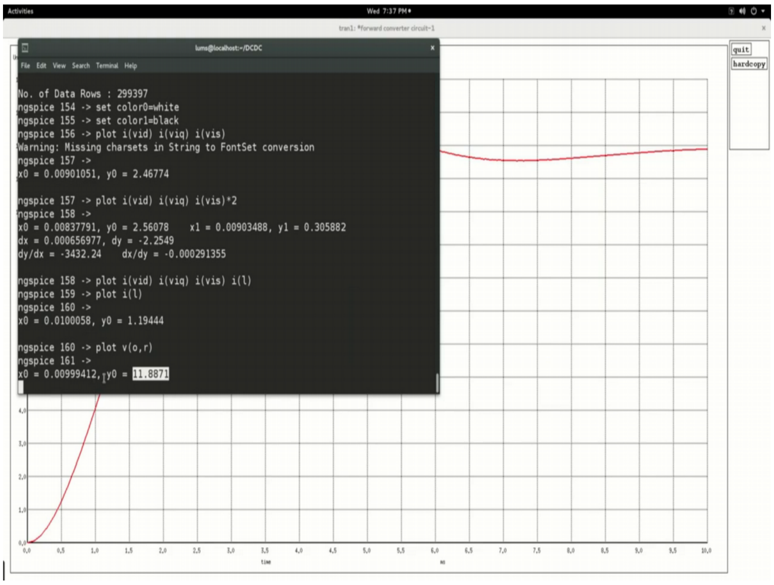
So, this will set the initial conditions see, it will be valid only for only for this value of inductance and state values.

(Refer Slide Time: 28:05)



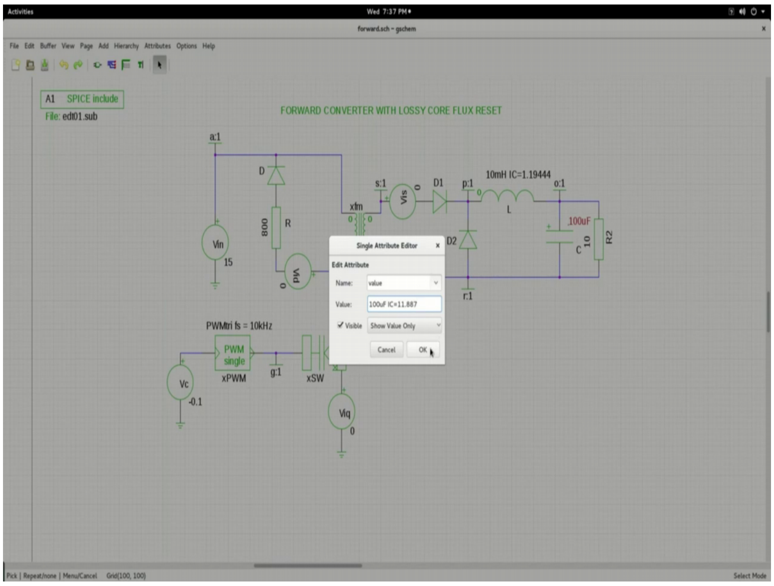
And next, let me see the value for the output voltage for the voltage across the capacitance plot V O comma R.

(Refer Slide Time: 28:21)

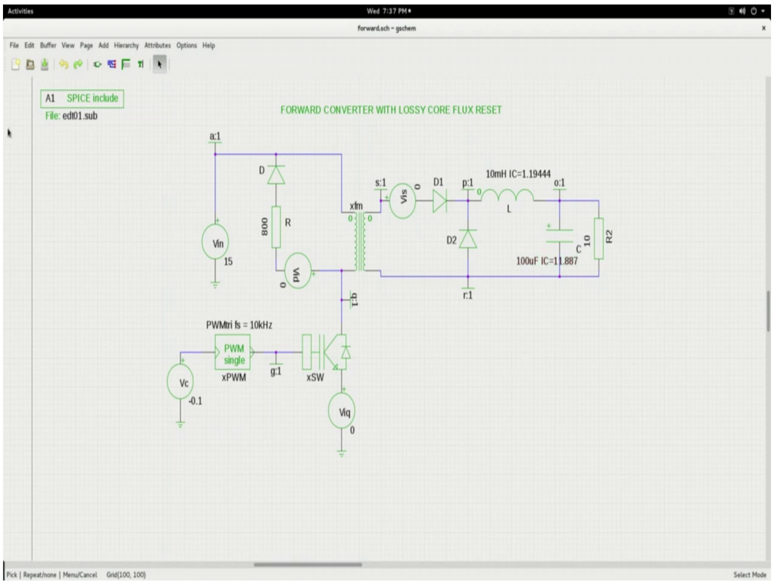


Now, let me take the end value here click on that. So, it is on 11.8871 there could be some minor difference does not matter, it will reach stable state quickly. So, that is 11.887. So, go to the value of the capacitance and give initial condition equals 11.887.

(Refer Slide Time: 28:43)

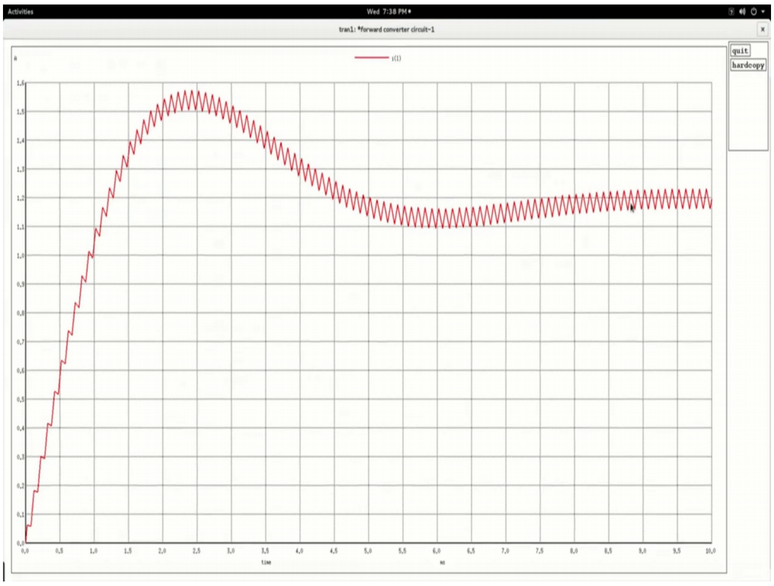


(Refer Slide Time: 28:59)

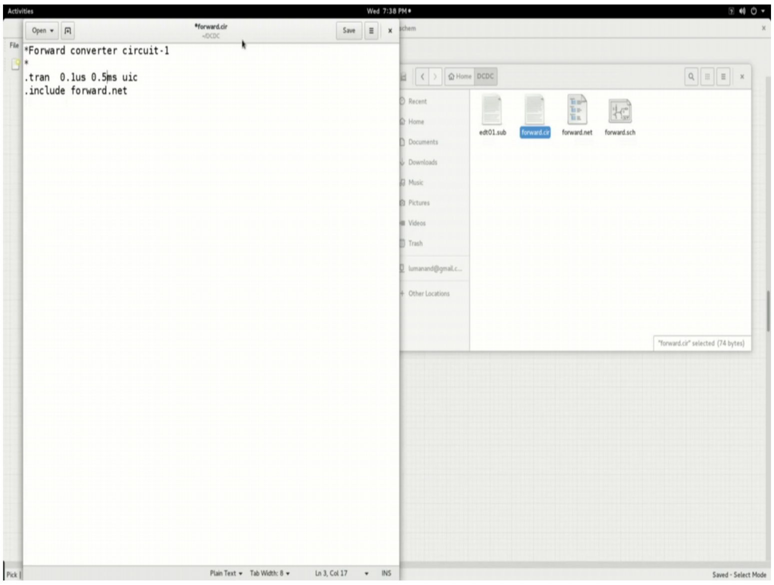


So, this is for the capacitance and let us save this and let us close and redo the simulation. So, now, before redoing the simulation I do not want to do the simulation for 10 milliseconds.

(Refer Slide Time: 29:09)

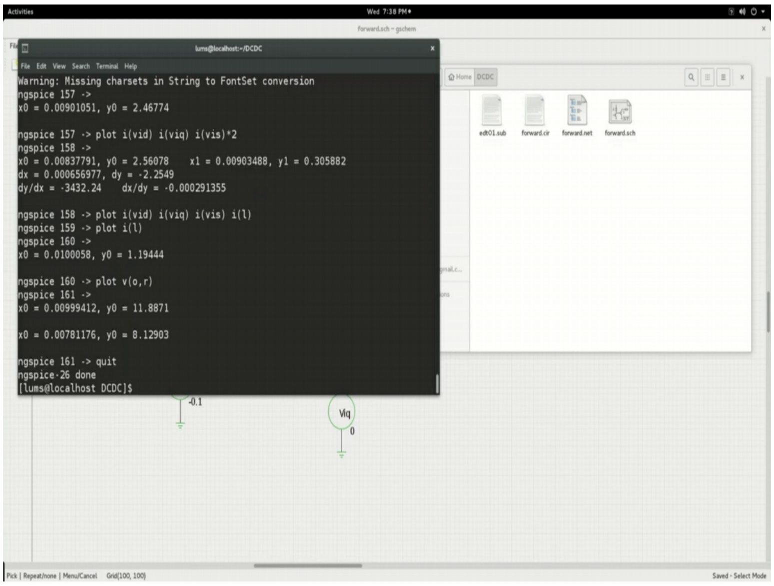


(Refer Slide Time: 29:17)

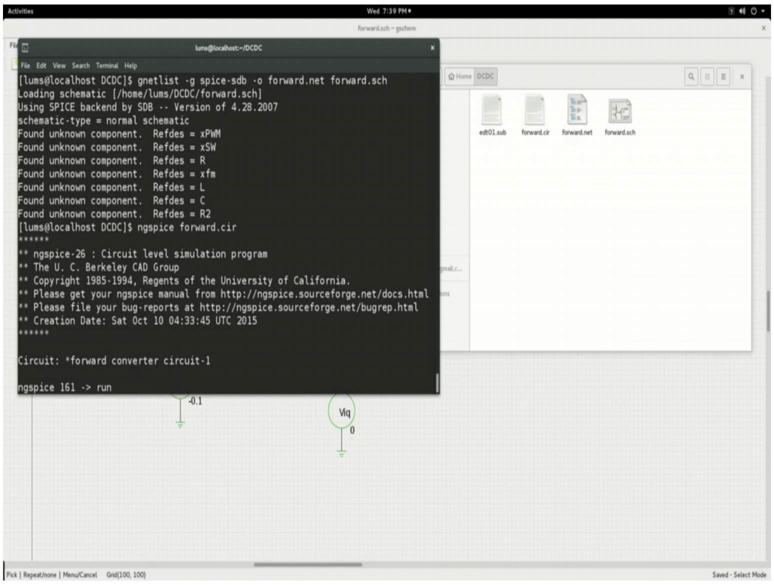


Now, that I have almost reached the stable state I will do it for let us say 0.5 milliseconds it is supposed to be pretty quick. Now rather than waiting it is actually 120th. So, therefore, let me save it and go to (Refer Time: 29:45).

(Refer Slide Time: 29:45)

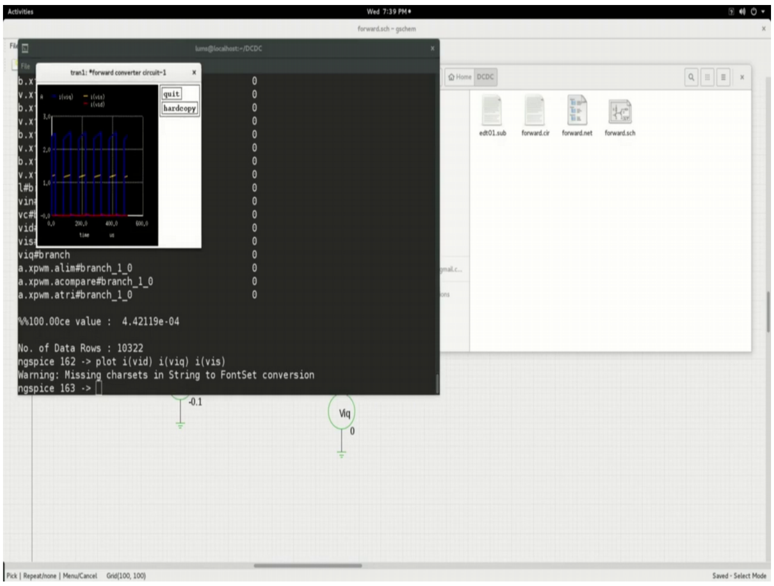


(Refer Slide Time: 29:47)



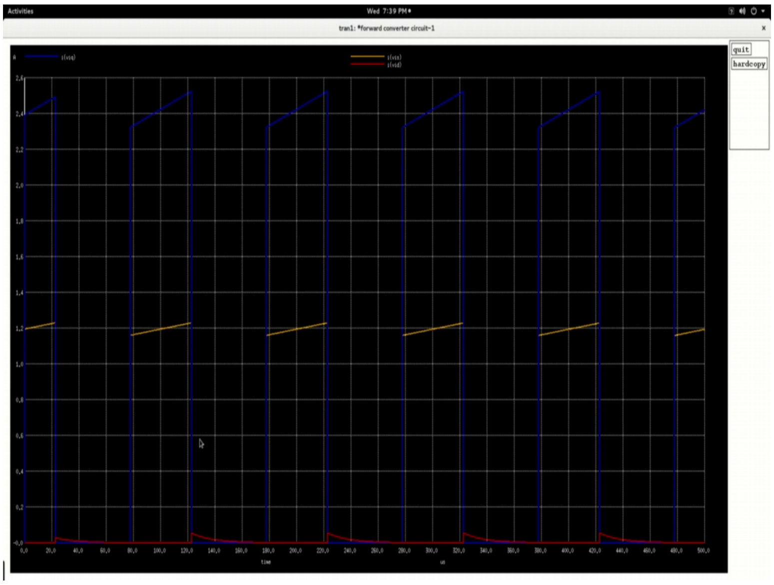
So, now let me redo the netlist. Now after doing the netlist Ngspice forward dot CIR, yes. Now, run the simulation.

(Refer Slide Time: 30:09)



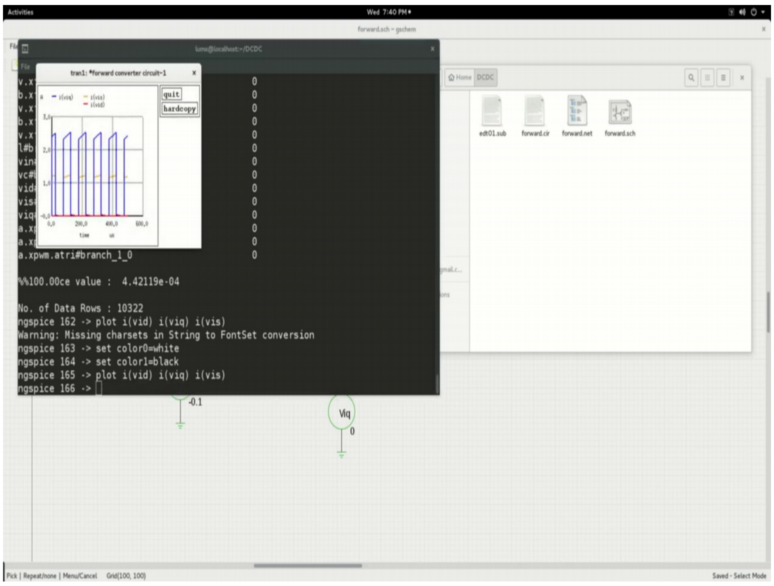
You see it is so quick and let us plot, plot I will plot the currents v vid plot the current switch current plot the current of the secondary. This is, ok.

(Refer Slide Time: 30:37)

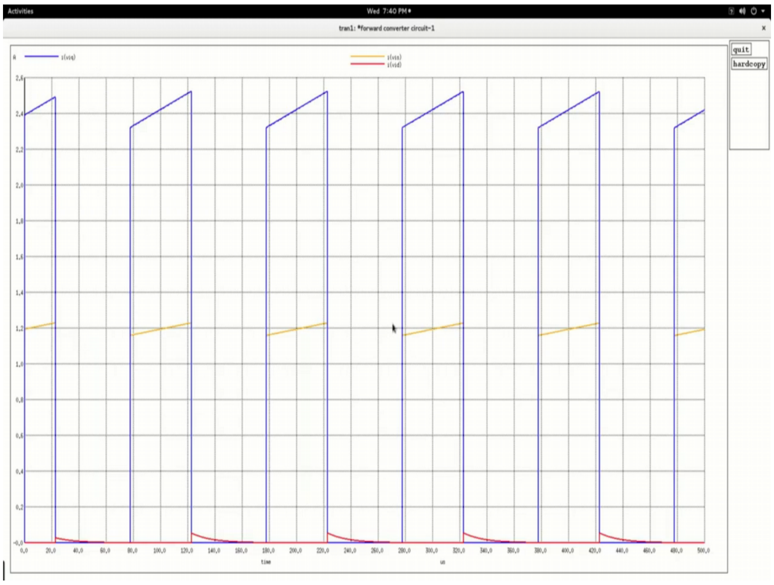


Now this is again the black background I do not like it, I will set color to white set color of foreground to black.

(Refer Slide Time: 30:43)

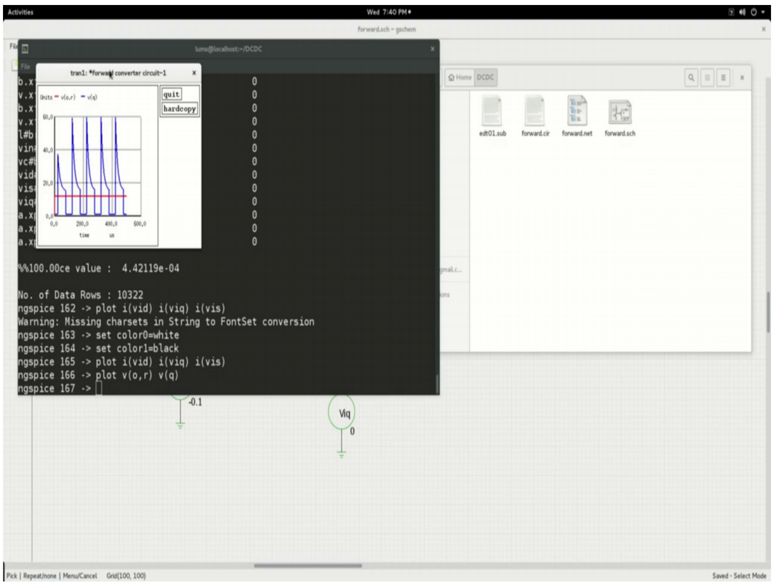


(Refer Slide Time: 31:01)

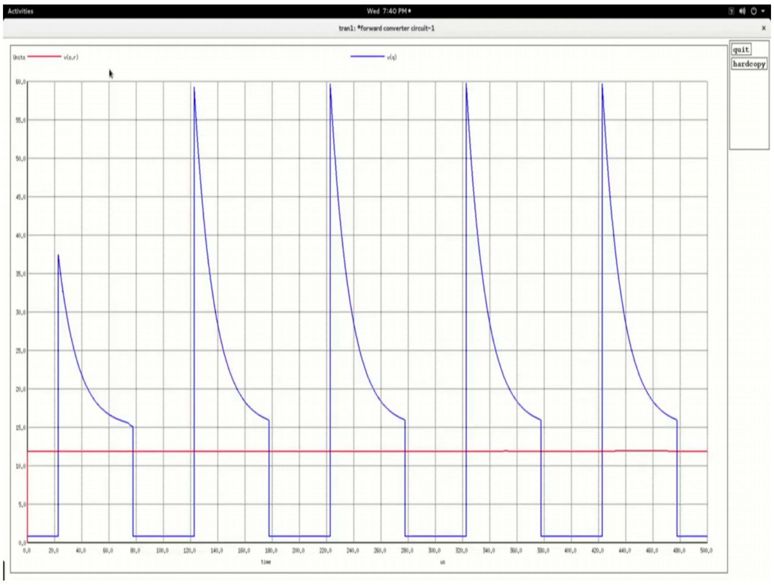


And look at the waveforms once again. And you see that you, you are already in the stable state. And you could also see the output waveforms V O comma R and probably you could see Vq switch voltage across the switch and distinctly you will see this is the output voltage V naught this is a voltage across the switch.

(Refer Slide Time: 31:07)

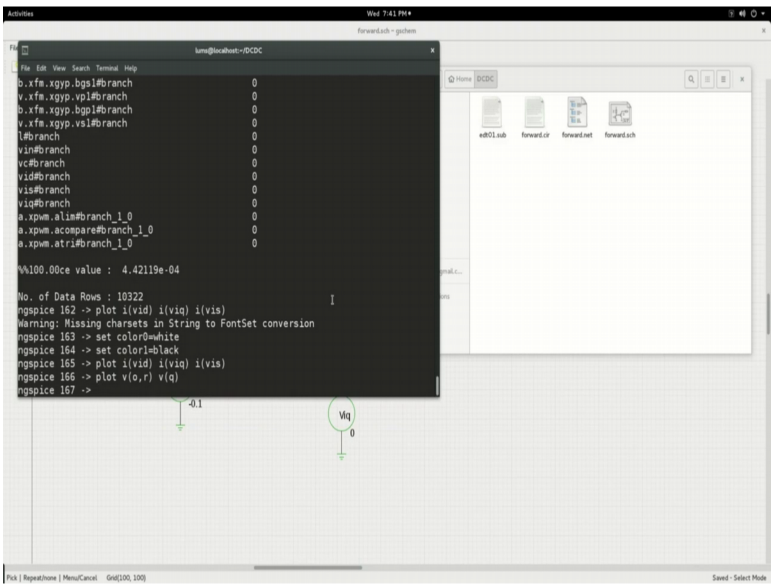


(Refer Slide Time: 31:25)



Now, this first one is a bit off, because our initial condition it will take some time for it to stabilize. It has taken just one switching cycle to stabilize and you see you get stable output waveforms. So, now, you can do lot of changes in your output load and the such accept any change in the state values which you have to run the simulation right from 0 to 10 millisecond. Once again and then re edit the initial conditions, but this stick this trick would be very useful. When you want to do repeated simulation of the circuit for changes other than the state values and you can quickly see the output results in this fashion.

(Refer Slide Time: 32:25)



So, I will leave it to you to explore the circuit and try to get as much insight as you can into this forward converter operation and the waveforms.