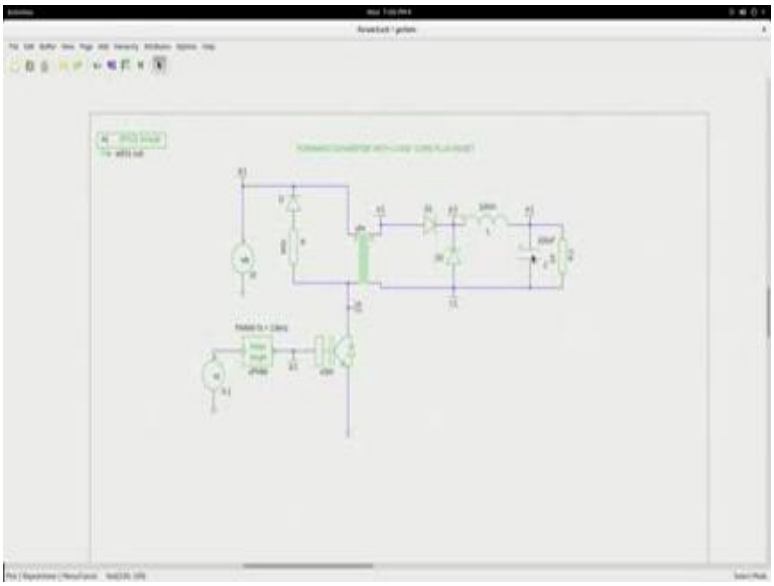
Design and Simulation of DC-DC converters using open source tools
Prof. L. Umanand
Department of Electronics System Engineering
Indian Institute of Science, Bangalore

Lecture – 14
Simulation of Forward Converter

Let us now perform a Simulation of the Forward Converter circuit that we have studied till now. I will open the DC-DC folder and this folder I have already written the schematic for the forward converter in the schematic file, I have also created a forward dot cir file and the edt 0 1 dot sub file which contains our custom models. I have also updated it to include the new custom models for the transformer that we will be using. So, if we open this forward converter schematic.

(Refer Slide Time: 01:05)



So, you will see on the screen the forward converter with lossy code flux reset calling it lossy because this is the core reset circuit which we studied and this is the lossy register which is going to dissipate the magnetizing energy of the core. This is essentially the forward converter circuit that we have been discussing till now. This is the power semi conductor switch which is going to turn on and off and it is getting right signal from a PWM block. And the PWM block is given with a control reference of minus 0.1.

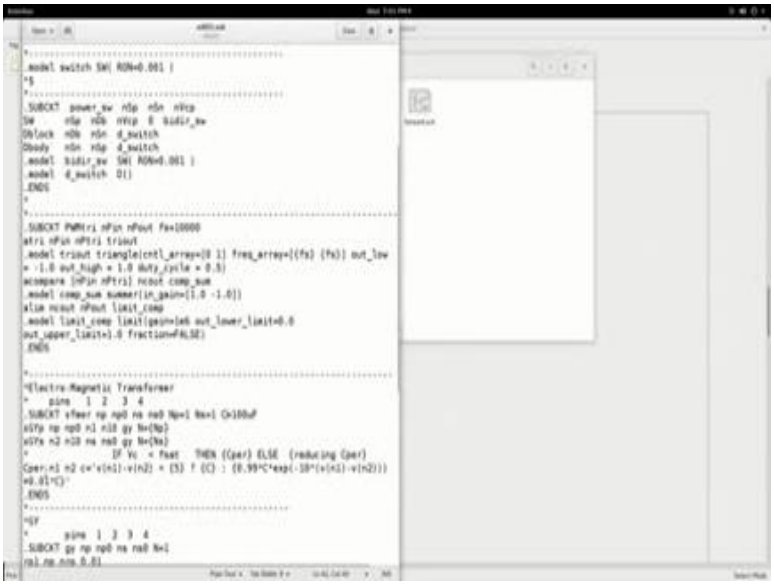
Remember last time when we are discussing the PWM block we said that this triangle within the PWM block is transiting from minus 1 to plus 1. So, at 0 if the control signal is 0 you will get 50 percent duty cycle. If the control signal is minus 1 then you will get 0 percent duty cycle. If the control signal is plus 1 then it is 100 percent duty cycle that. 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 you come to the transformer portion, the transformer is not the regular transformer provided by spice that is the linear transformer model. We have included a sub circuit model here so that even saturation effects can be taken care off. 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 prevailing circuit, we have register and the diode. And this is of course the input and we are applying 15 volts DC here.

On the secondary side we have the diode D1 and D2 and L C 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, 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. Let us say we have reference node R here. So, we will call this the reference node and anything on the secondary side we will measure it with respect to this reference node. When you want to see the secondary voltage you will say V S with respect to R you have V P with respect to R that is V P comma R, V O 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 use for the buck converter and you can use the similar thing for the forward converter example here two, using the PWM with the switching frequency of 10 kilo hertz. 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 profile let us simulate.

(Refer Slide Time: 05:42)



Now, going back to the folder the forward dot cir; the forward dot cir contains a dot transient analysis statement 0.1 micro second step and then up to 10 milliseconds use initial conditions command. 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 edt dot sub I have already included the model for the electromagnetic transformer.

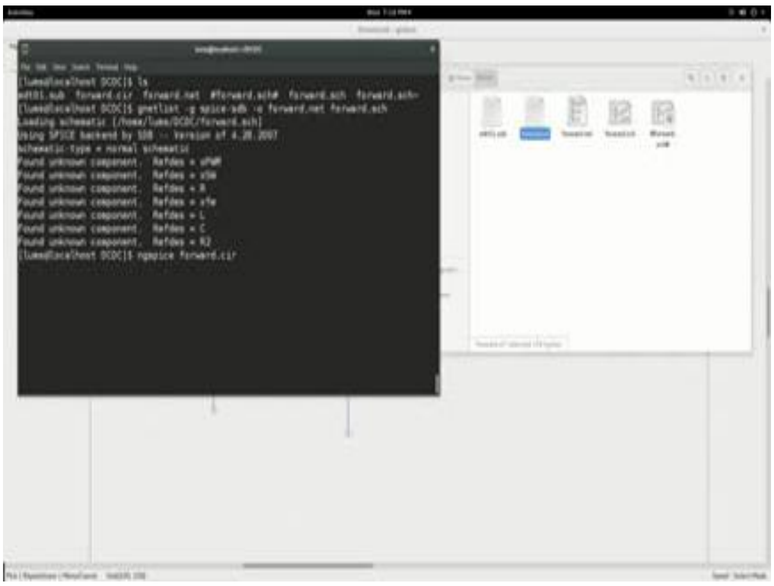
So, there is a model for the electromagnetic transformer the primary dot point, primary non dot point, secondary dot point, the secondary non dot point, there is a default primary number of turns secondary number of turns and C. Here C is not to be thought of as capacitance, but as permeance. So, within the magnetic domain the permeance in the core behaves very much like the capacitance or capacitor. So, in spice the equivalent component would be C and therefore we have to C. I will explain more about this capacitance permeance relationship later on when we are discussing the magnetic.

So, we have used this kind of the model here we have also put some kind of saturation effect here. If the value of the flux is greater than a particular value then 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 a decaying down to 0. This will emulate the b h curve of core. So, this is a saturating type of a magnetic circuit and therefore this generic transformer which behaves much more closely to reality.

Now, this transformer uses two further models called gyrators, and the gyrator is modeled here. Anywhere this is the model of a control magnetic transformer and you will find many models of transformers in 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 broadly state the concepts. Only if you want to look at saturation then you will have to include something to make the core saturate.

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 Ng spice engine which is what we will be doing.

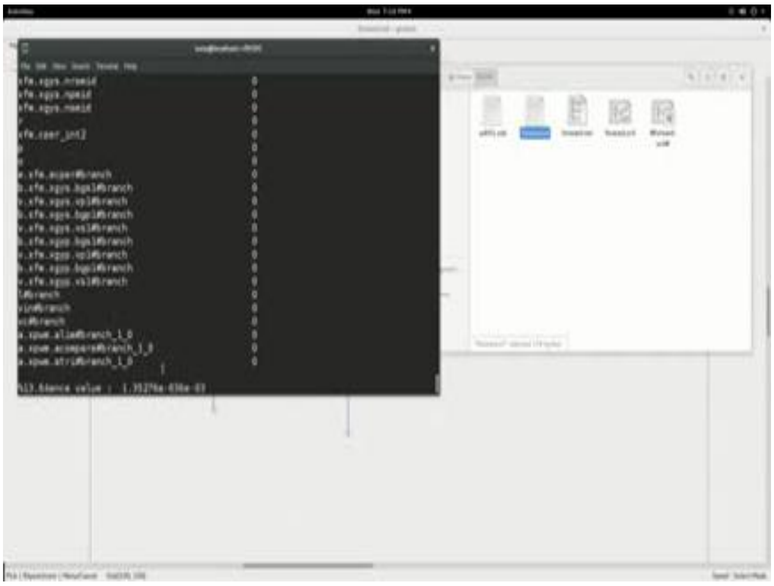
(Refer Slide Time: 08:55)



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 net list and also call Ng spice after we have generated the net list. Let us now generate the net list and go forward. So, generating the net list is with this now familiar command g net list dash g spice dash sdb 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 will see that there is a net file generated.

Now, we can use this net file this, net file is being called the forward dot cir and Ng spice can perform the simulation. Now on the Ng spice engine Ng spice forward dot cir.

(Refer Slide Time: 10:04)



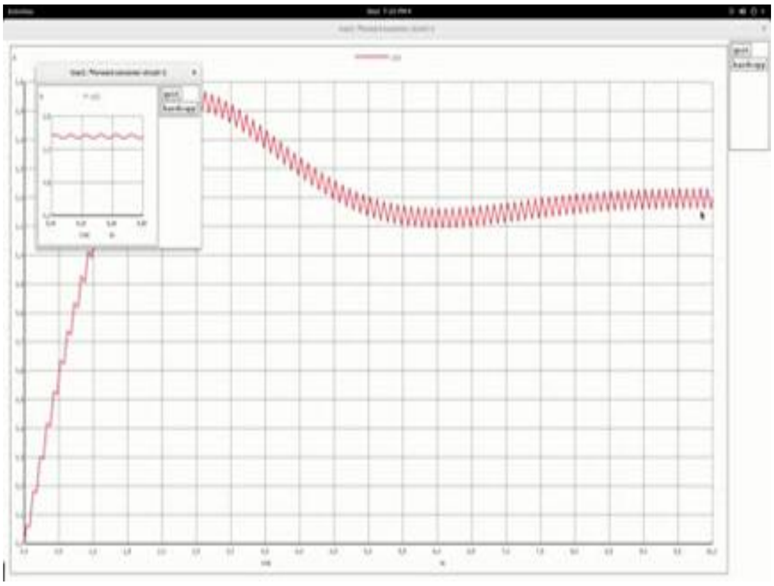
Now, this will load the circuit schematic with the net list and now run to perform the simulation. So, this will take some time. I will later on tell you methods in which you can reduce the simulation time. The first time simulation will take quite some time basically because you would have given an 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 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 75, 80 percent of the simulation done, you just have some more moments to wait and you have the complete simulation done up. So, you could plot. When you plot, let us say I want to plot the output do not just put V in brackets o. Now with respect to 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. Coming back here you will see well this is the output wave shape.

Now, let us before we further investigate few more aspects of the wave forms of the other parts of the circuit. One tip here would be, let me put that back here you have a black background and on the black background you have the white grid and the wave form showing up in color here. In some cases you would prefer to have a white background and have a black grid lines and at the wave forms and color. This is especially useful when you want to document it on cutter we take a screen shot of the brought out result and then put it into a document.

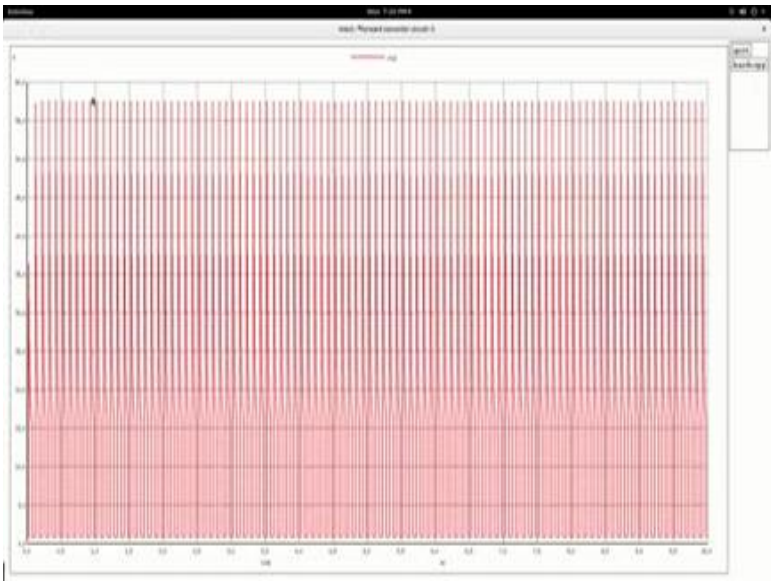
So, to change the background what you could do, you could use this following commands you can also always go back to the Ng spice manual and check about these commands. Color 0; so color 0 I will now set it into white. This basically sets the background color 0 represents background color. Set these are fixed it reserved words, color 1 is the foreground which we use again to black. Now, the same plot command would appear in the much deserved black white background with black foreground.

(Refer Slide Time: 13:43)



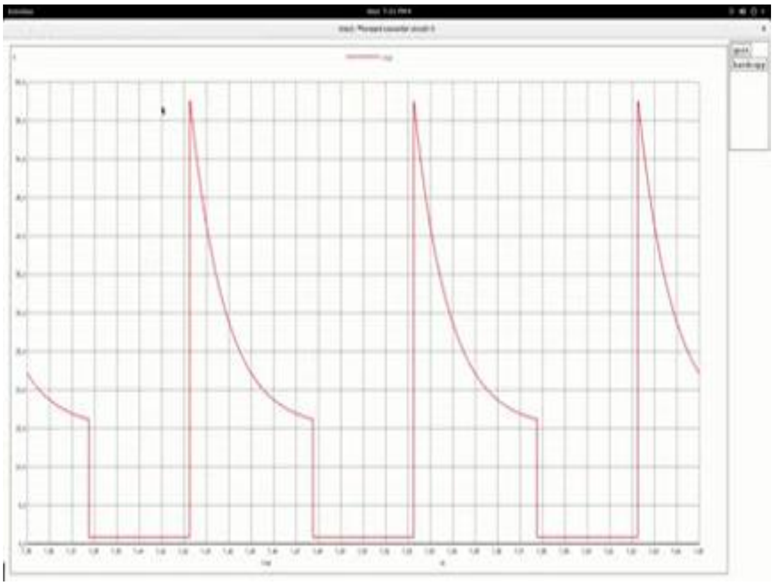
Now with this let us see few more wave forms of interest to us. We would like to see the inductor current wave form here. so let us have a look at the inductor current waveform plot i l, and see I have expected the inductor current triangular wave shape this is what we are expecting. And you observe 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 to do to short term the simulation time, if you are not interested in the trenchant.

(Refer Slide Time: 14:41)



Now, another point that we would like to see is v q here, we have been discussing so much about v q the voltage across the transistor. So, v q let us expand it to just new few cycles and you would see.

(Refer Slide Time: 14:52)



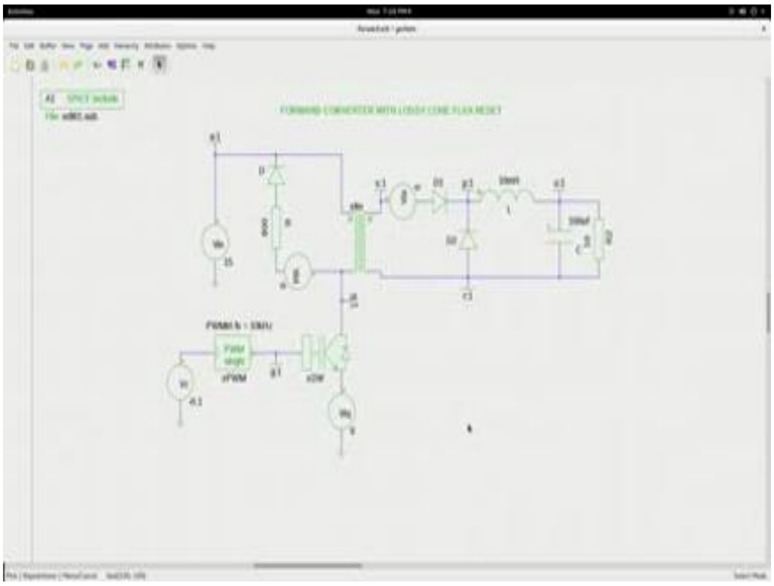
Now you see this is exactly what we have been discussing this is the Pontian, so therefore on state. Then the moment the BJT switches off so you see a high spike and then the current decades exponentially and goes towards V in value or VCC value. So, this is typical of the lossy flux this right hand. This exponential d k is the l m i r time constant and this is due to the decay in the magnetizing current. So, this is one important wave form which I wanted to show you.

Another point important point of wave form 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 being a state or energy storing variable component the current is state variable and it will also 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 observe that the branch currents are available in the list of plot outs only for l branch if there is a source V in and Vc branch.

So, you see that there will be currents available only for the source branch another source branch and n l branch. So, how to see a measure currents at various other points? So, I will give you 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 if we can do such a thing and the meantime first what I will do is quit this plot and clear the screen and now we will do the modification here.

(Refer Slide Time: 17:50)



So what I am going to do? The source are like this I am going to introduce here, here, 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 this circuit schematic.

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

Otherwise there is no other change that we have done. We have now put current sensors and with these current sensors we will be able to get more inside into the circuit. So, let me save this and do the simulation once again.

(Refer Slide Time: 19:10)

So, let us generate the net list, net list is generated, and now we will go into Ng spare of spice and run the simulation once again, so it should take some time.

(Refer Slide Time: 19:30)

And 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.

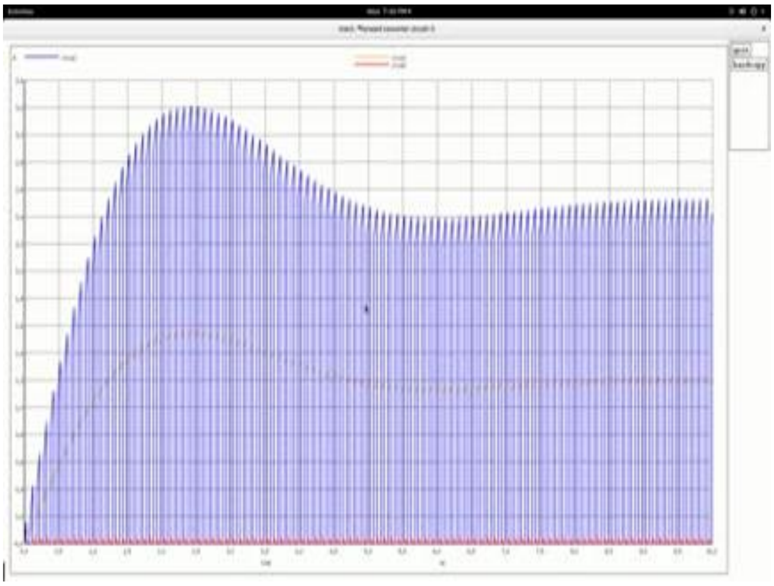
Now, while the simulation is going I would like to also show that in the transformer when I double click on the transformer it is a sub circuit I have put here the primary number of turns 10, secondary number of turns 20 and the permeance is 100 micro. So, this is the parameters that are passed on to the transformer. Therefore, you see that n p 10 and then n s 20 means there is a turn ratio of 1 is to 2.

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 to 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 be also be a magnetizing part, so these two components together form the switch current. And during the time the switch is off the reflected part is not fair 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.

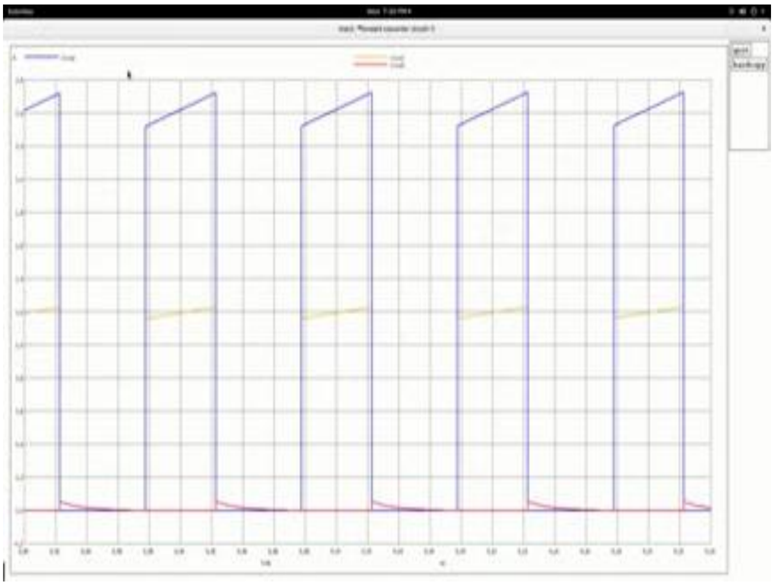
You could see here you have the vid branch current, vis branch current, viq branch current. So, voltage of the demagnetizing winding, divert winding dit i of the current flowing through the switch viq and the current flowing through the secondary vis.

(Refer Slide Time: 22:17)



Let us plot that, let me expand this, and you will see that let me also cover the see just few here.

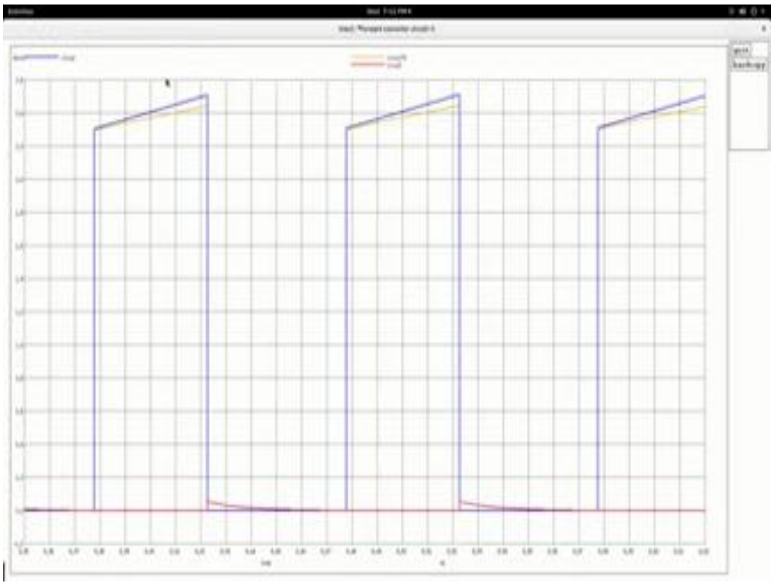
(Refer Slide Time: 22:26)



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. We call that this point here where the cursor is would be n times i naught. Now this is actually i naught current, so if I plot inductor current it will flow all along through like this. Therefore, it may be good idea to plot that out also I will do that later. This is actually the secondary current. And you see the red one is the demagnetizing current. See it is so small in magnitude, so actually magnetizing art will take a linear wave shape here and goes in a exponential manner and from here again linear and so on.

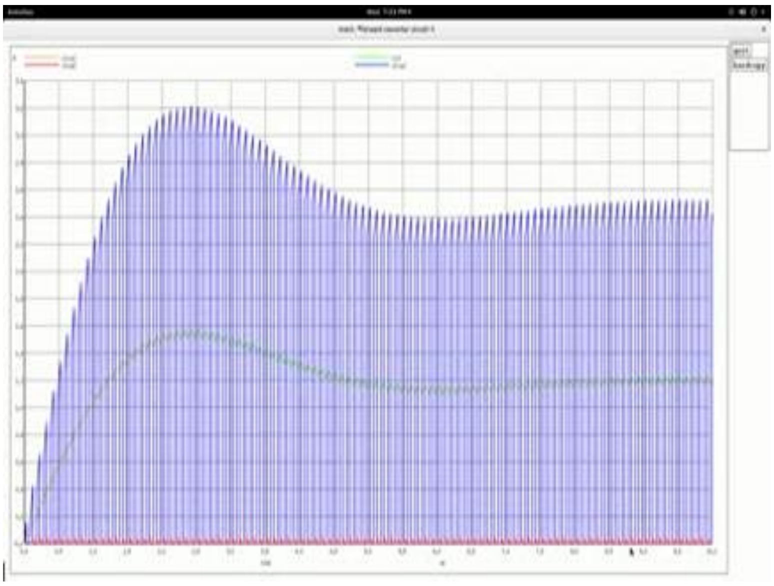
So, let us say if I say n times i naught this is supposed to match here exactly except for the magnetizing. You will see that this will more or less come in this like this here, like a a small triangular gap be left here that would be the magnetizing portion. So, let us see that. What I will do is, now I will give the, this is actually the secondary current into n and n is two in this case, and now plot that and let me expand that, now you will observe.

(Refer Slide Time: 24:35)



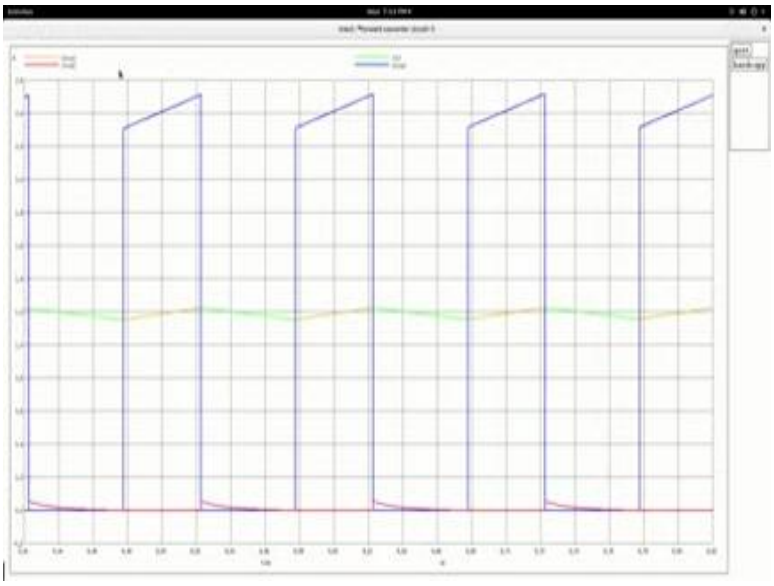
Now you see this the primary and the secondary are exactly same except for the difference which is the magnetizing part. So, the primary switch is composed of the reflected part and the magnetizing part. And the magnetizing part actually is an amount equivalent to this which will come and match exactly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

(Refer Slide Time: 25:28)



So, that is one aspect I would also like to compare with the inductors wave form. So, let us say I will remove this and also put i or l the inductor wave form. Interesting to see that let me zoom in.

(Refer Slide Time: 25:33)



Now you will see the green is the inductor wave form. You see that this inductor wave form 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 though the primary. Now, the central say I have the average of the inductor wave form is i naught and that would come as the flat top equivalent of the primary switch current which is n times i naught. So, the fly primary switch currents flat top equivalent, equivalent flat top current will be n i 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 repeatedly some changes and check the wave form 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 of one thing what you could do is, you could take the value of the state variables. You will see there are two state variables; the inputs and the state variables defined system. And inputs are DC in this case and state variables are i l and Vc.

So, let us take the value of i l 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. That is one exercise I will just show you; I will just quit from here no not quit I will just plot, plot first i l 1 state variable. So, let us take value at the end of the simulation here click on that and then you see the values one point at the 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 (Refer Time: 28:08) I will also put initial condition equals 1.19444. This will set the initial conditions, we valid only for only for this value of inductance and the state values. And the next let me see the value for the output voltage across the capacitance plot v o comma r.

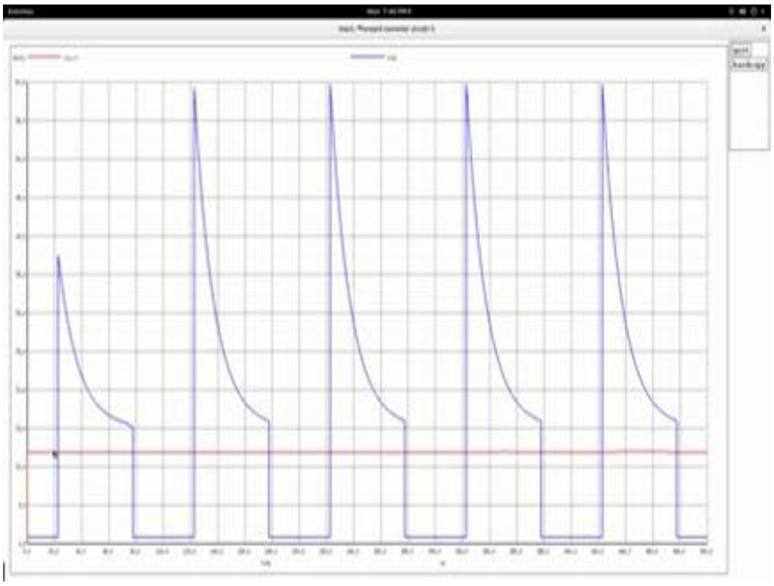
(Refer Slide Time: 28:52)

Now, let me take the end value here click on that. It is around 11.887. There could be some minor differences 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 11.8871. So, this is called capacitance and let us save this and let us close and redo the simulation.

Now, before redoing the simulation I do not want to do the simulation for 10 milliseconds (Refer Time: 29:51) now that I have almost reached stable state I will do it for let us say 0.5 milliseconds then it is supposed to be pretty quick now rather than waiting it is actually one twentieth, so let me save it and move to control n. Now let me redo the net list. Now after doing the net list Ng spice forward dot cir, yes. Now run this simulation you see it is so quick and let us plot, I will plot the currents; v v i d plot the current, switch current plot the current of secondary.

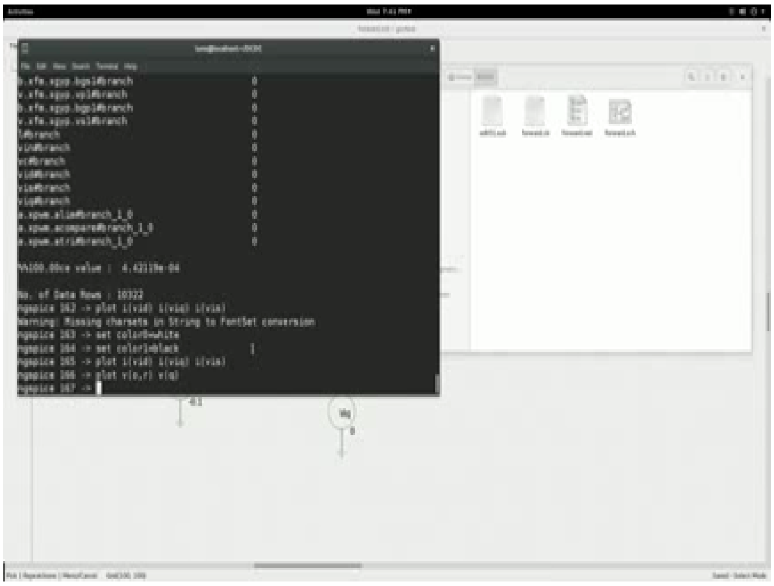
Now this is again black background I do not like it, I will set color to white set color of foreground to black and look at the wave forms again and you see that you are already in the stable state. And you could also see the out of wave forms we o comma r and probably you could see v q switch, voltage across the switch.

(Refer Slide Time: 31:59)



Interestingly, we will see this is the output voltage V naught, this is the voltage across the switch.

(Refer Slide Time: 32:57)



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 a stable output wave form. Now you can do lot of changes in your output load and search except 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 tip this strict 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.

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 wave forms.