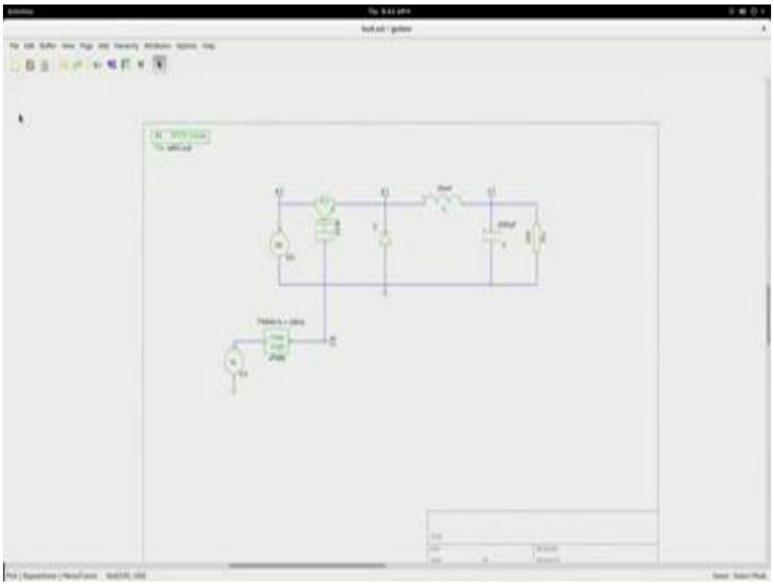
Design and Simulation of DC-DC converters using open source tools

Prof. L. Umanand
Department of Electronics System Engineering
Indian Institute of Science, Bangalore

Lecture – 09
Simulation Example of Buck Converter

Let us begin the simulation of the buck converter we shall move to the applications start up the g d s schematic, (Refer Time: 00:29) schematic capture you make. Let me go here type first save this I will put it into d c d c e and let me call it as a (Refer Time: 00:44) d c d c folder and we call this one as buck. So, we have the dot s c h file here.

(Refer Slide Time: 01:45)



So, now we will start loading the components and like before I will go into the components we need (Refer Time: 01:07), we need voltage force not sine wave this are the default we will change that d c voltage source, we will need the diode, we will need an inductor, we will need resistor for the load then we will need a power switch, we need power switch control switch.

So, this is a generic control switch could be an I g b t or (Refer Time: 01:54) b g t. Now to control the drive for a give the drive for the switch we need to take a block for here go the a block like the (Refer Time: 02:12) block library and take P W M single, as it is only one switch that we need to control and then let us pick one more source block for giving the input to the control element and that we can take from even these groups then, we need the labels, we need the power rails. So, the generic power symbol this is for the labels we picked up one side we need to label the various parts we need the ground symbol of course, very very important you cannot do a spice simulation without ground symbol, and then we need to include the directive, include directive. We call that we would like to include our custom library which will now start growing it has two components diode and the micro diode model.

So, in place (Refer Time: 03:37). So, these are the components that we have to deal. So, let me first give a label name for that and then we shall go to save it as to refer to e d t zero one dot sub. The same name that we gave for rectifier and to that file will include or add few more custom models. So, let us place now this is a d c source, which would not be a sin. So, let us keep the d c source value (Refer Time: 04:19).

And this one will be v in. So, we have this, input source and this is also a d c source. I will plot now with this as zero point five volts and call this one as b c r the control voltage that you would give for the p w m. So, let us call that one as v c, which we need to be doing that input of the P W M here. Let us organize the, now for the buck converter that will try to rotate it and then place it here and then we need to have a diode rotate that and then lets place the diode here, then we need to have a inductor place it line. So, that (Refer Time: 05:33) connection wires let us rotate load resistance for path place it here, and the P W M somewhere here and this. So, these are positions that we can make a copy of ground signal and paste it. So, that you have one more ground signal here. Now, let us next step is to make the connections.

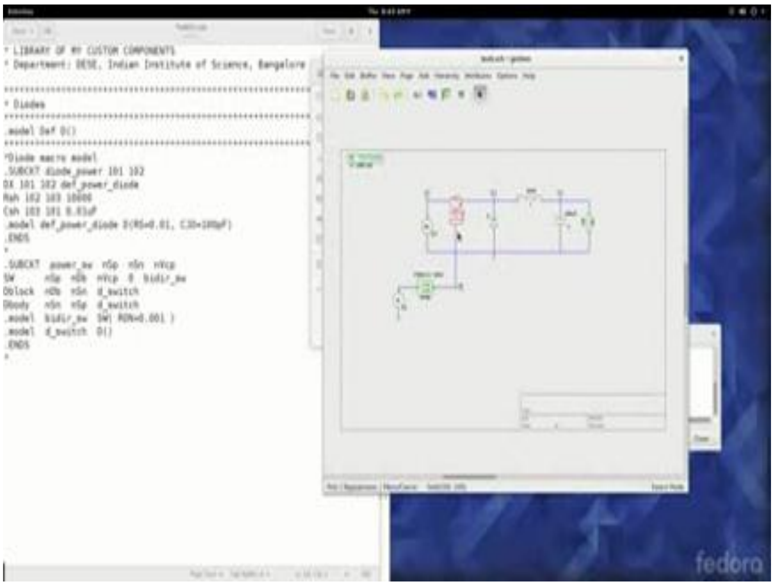
So, this gets connected here. Then we need the inductor diode to the pole, to the capacitance to here (Refer Time: 07:04). Let me drag it make a connection make this connection here and this has to be connected to this. This would be connected to this and we have almost all connections observe that here it is just over lapping and not making the connection the spice has not got an node point here, but if it puts the node point then it is connection be careful that should not happen. So, we shall put a ground node point there and we will put the ground node here. So, I will leave the switching frequency default at 10 kilo watts which would be in 100 micro seconds as e s switching to here. Now, let us put some nodes here. So, now, this is my v in node.

So, I will name it as a, I will this is our pole node. So, name this as p as we had discussed the output form. We will name this as O. Let us have the gate drive node also to see the, how the gate drive looks like and that we will call as g the gate drive node and of course, in all control voltage V C. So, if we look at the gate drive node here, here you will get the pulse with modulator signals, pulse signals which will be controlling the on off of this and how the power gets pumped into the inductor and the output capacitor of the load.

Next, let us put in some proper labels and we say this is switch there is only one switch I have not named it as x w 1. x means that this is the sub circuit x P W M is the sub circuit. So, that is named and d only one diode maintained that 1 l. So, no (Refer Time: 10:31) number 1 c and this is our r naught. So, let us I will give some values gate 1, I will show you how to calculate values for now I will put in the values here 10 million 3 thousand microfarad hundred ohms at the output and the 10 kilo hertz switching frequency and you have point 5 volts as a control voltage here.

Now, this will complete the schematic part of it. We need to put in the model for this and put in the model for this need to the sub-circuit. These two are the components is a sub-circuit the generic model for this will coming back and how the P W M is made P W M is a analog behavior of model and that also will come in here n g spice is very powerful it also permit's analog behavior model where you can give logic and analog symbol processing and interface that digital and analog mixed (Refer Time: 11:59) is also possible. Very very powerful spice (Refer Time: 12:04) to have. So, let us now go and save here good time to save.

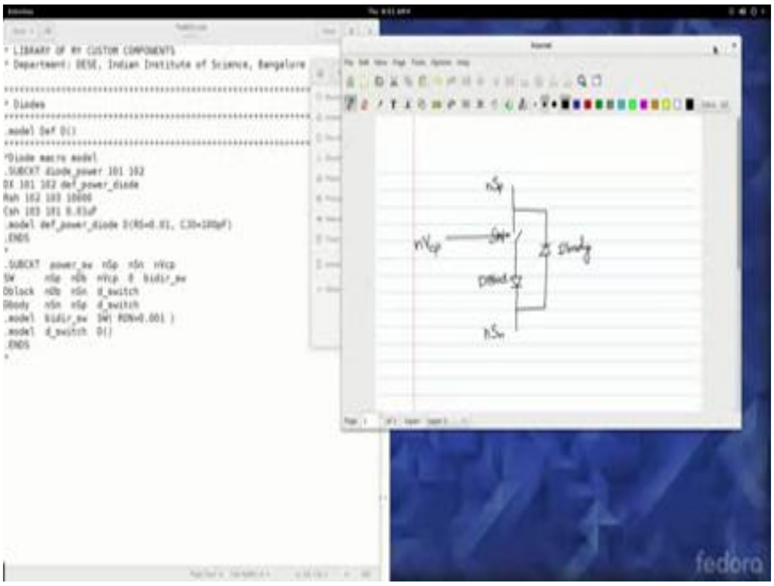
(Refer Slide Time: 12:37)



So, this is the buck dot s c h which is there in the d c d c folder here and I have copied the e d t 0 1 dot sub from the rectifier example that we did in the last week and placed it here. This contains exactly the same content as that possible. Now here we will add two more one is for the switch and other for the P W M triangle. Now for the switch I will put in these lines. These are the sub-circuit, this is the power switch, the power switch which is there in the schematic that we explained (Refer Time: 13:03).

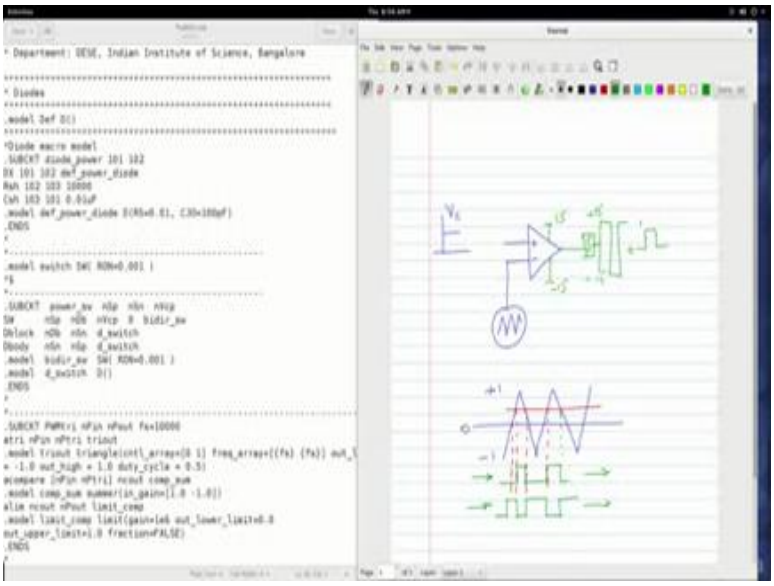
So, this is the power switch. When you double click on that you will see that coming up here the value p. It has a positive node and the negative node that is what this is positive n s, n is the negative and v c p is the control gate. Now it uses a switch and do diodes to model this particular power switch. How it looks like is as follows. Let me open up my writing pad it goes like this.

(Refer Slide Time: 13:51)



So, I have that switch, this is the switch and let us have diode like this and a freewheeling body diode in this manner, these we forming it as n s p. This is called n s n and there is control in n v c p. So, this is blocking diode it allows only the forward path as far this switch is concerned and any freewheeling action goes through the body diode which is d body. So, this is essentially what goes into a makeup this sub-circuit. So, let me close this and come back to this. Now here if you see there is a diode model, the diode model already we have defined and rectified, we do not need to define again. There is a switch model which we need to define. This is using the using (Refer Time: 15:13) model command of spice. So, switch is also recognized by spice but let us indicate here. So it goes like this point.

(Refer Slide Time: 15:01)



So, now we have the switch model and sub-circuit power. And we have the power switch well defined. Let us save this. Next we have to define the p w m. For the case of P W M block we shall do it this way let me put in the text for the model. So, you see here this is sub-circuit P W M triangle n p in and n p out and there is a parameter which is switching frequency which you can change and will lie for different circuits. It has these arts I have used a as a prefix for these lines here to indicate that they are analog behavior of modeling, not necessary that if you choose a, but it is a good practice.

Now it has three major parts which I will show by going to the data. So, now, what is done is; there will be comparator and to be comparator arms let us say you have plus and minus. You will give a triangle generator here, and this is where the comparison input for the control voltage is time and this would get compared with the triangle. So, if I am having a triangle which is from minus one to plus one so zero in between and let us say my control voltage v c e cuts in like this.

So, you would (Refer Time: 17:30) have here also in modulation which looks like this and so on. So, it will compare with this control voltage and if the triangle is greater than the control voltage you will get a high pulse, the triangle is lower than the control voltage will get a low pulse. This in one way or if the control voltage is higher than the triangle you would get the (Refer Time: 18:18) shape like this, and whenever the control voltage is higher than the triangle it coming out this. So, whether it is this or this output would depend upon your plus or minus that you set for the comparator, but you can generate at well the suitable time for P W M pulses for this kind of a configuration.

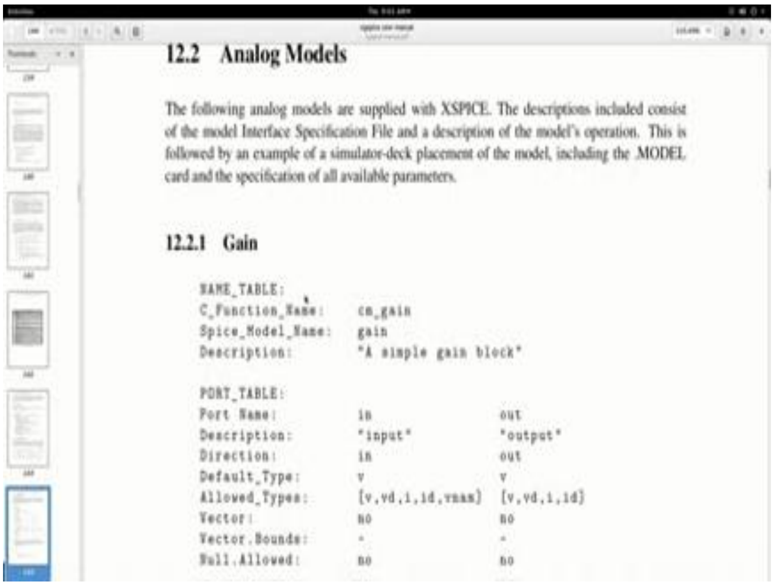
This is the (Refer Time: 18:53) what that we are doing here. So, we have triangle generator just like what you have shown here. We have a comparator, a compare and then a limiter. What happens is at the output of this will stream to plus fifty and minus fifty to the positive and negative range. So, you will get most of minus fifteen into plus fifteen. We need to limit it to such that it goes 0 to 1, 0 to 1. So, normally we put a limiter here and this limiter will see that the output will from 0 to 1.

So, this is exactly what we have been doing. So, we have a triangle generator and compare and it is model and limiter the output of which you see that the lower limit is 0 upper limit is said that 1. This unlock behavior modeling is interesting you should look into the n g spice Mangalore, it is very very elaborate and this specificate in detail how to analog behavior model. (Refer Time: 20:14) into the n g spice user manual and in the chapter mixed mode and behavioral modeling with x spice, x spice is integrated into n g spice you will see the analog behavior on models given for various function blocks. And we have used the limiter, we have used summer these two blocks and then triangle generator.

(Refer Slide Time: 20:15)

So, if you go in to the analog behavior model read the complete description on each other's model are given and how to use them into your the sub-circle.

(Refer Slide Time: 20:53)



12.2 Analog Models

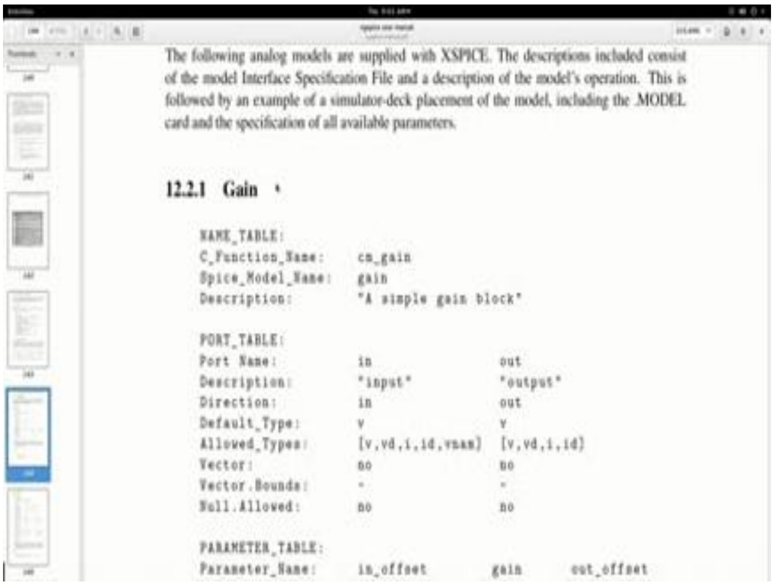
The following analog models are supplied with XSPICE. The descriptions included consist of the model Interface Specification File and a description of the model's operation. This is followed by an example of a simulator-deck placement of the model, including the .MODEL card and the specification of all available parameters.

12.2.1 Gain

```
NAME_TABLE:
C_Function_Name:    cm_gain
Spice_Model_Name:   gain
Description:        "A simple gain block"

PORT_TABLE:
Port Name:          in                  out
Description:        "input"             "output"
Direction:          in                  out
Default_Type:       v                   v
Allowed_Types:      [v,vd,i,id,vnam]    [v,vd,i,id]
Vector:             no                  no
Vector.Bounds:      -                   -
Null.Allowed:       no                  no
```

and see that here when it is (Refer Time: 21:04).

(Refer Slide Time: 21:00)



The following analog models are supplied with XSPICE. The descriptions included consist of the model Interface Specification File and a description of the model's operation. This is followed by an example of a simulator-deck placement of the model, including the .MODEL card and the specification of all available parameters.

12.2.1 Gain

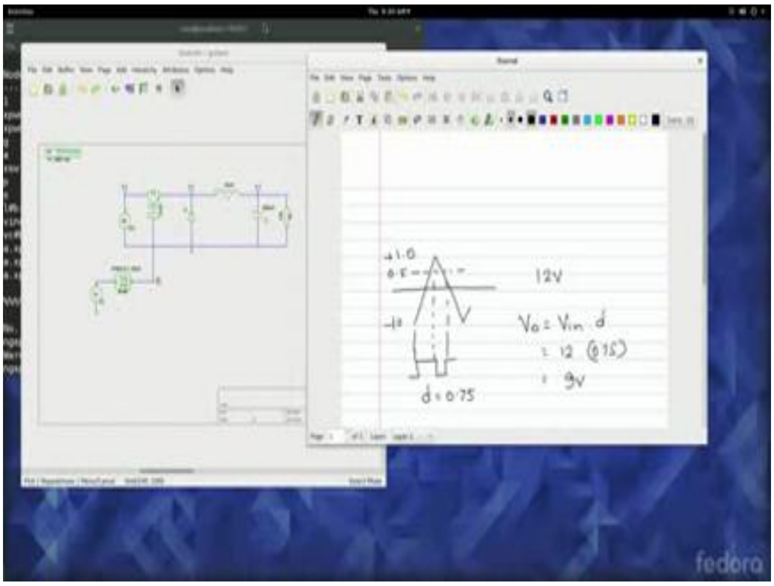
```
NAME_TABLE:
C_Function_Name:    cm_gain
Spice_Model_Name:   gain
Description:        "A simple gain block"

PORT_TABLE:
Port Name:          in                  out
Description:        "input"             "output"
Direction:          in                  out
Default_Type:       v                   v
Allowed_Types:      [v,vd,i,id,vnam]    [v,vd,i,id]
Vector:             no                  no
Vector.Bounds:      -                   -
Null.Allowed:       no                  no

PARAMETER_TABLE:
Parameter_Name:     in_offset       gain        out_offset
```

So, try to go through this part of the manual try to get much better insight into analog behavior model. Coming back to our e d t 0 1 dot sub, let us save and close this. So, this is included here. Next step that we have to do is have a buck dot c i r, that we can simulate that. We shall now open note pad and then save it as buck dot c i r in d c d c save. So, you have got that and let us enter the c i r statements.

So, let us write buck converter circuit leave one space dot ram that is micro seconds (Refer Time: 22:19) using conditions and dot include buck dot net then save this file. So, now, we have all the things in place, buck dot c i r, buck dot s c h that we have to generate the net list and then simulate. So, let us do that. You go into terminal c d 2 d c d c we have (Refer Time: 22:56). So, use the comments g net list as g spice dot spice dash s p b now to output buck dot (Refer Time: 23:09) s c h.

So, to do that in buck dot net is generated with a have a look at that. You see that this models which we have told let us utilize and then the net list comes in here, this is what we have (Refer Time: 23:28). So, now, time to simulate. So, in the n g spice buck dot c i r there is loaded run the simulation runs and everything is available and you can plot. So, plot let us say v naught. So, this is the product v output it goes and settles at around between 8 and 9 volts and then we have given I duty cycle of point 7 5; how did you give a duty cycle point 7 5.

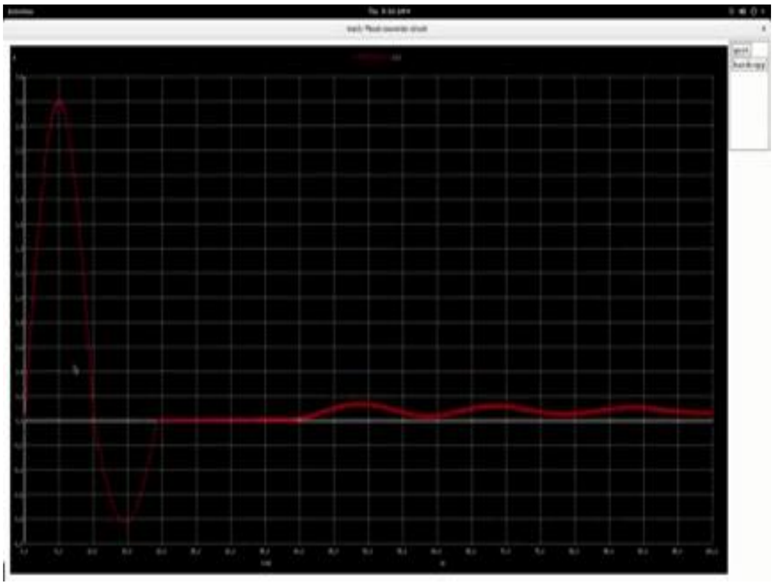
(Refer Slide Time: 24:37)



We call that in the schematic, you have given point 5 or c r and we know, that let me go to notepad. We know that we have the triangle which moves from minus 0 1 point 0 to plus 1 point 0. We have a given at threshold control voltage (Refer Time: 24:59) at time 5 volts. So, this would be 75 percent of a prime and therefore, you will have something like this. So, this will have d of 0 point 7 5.

So, this is how 75 percent (Refer Time: 25:20) come. So, 0 point 7, we have given an input voltage of 12 volts or v naught will be equal to v in into d which is 12 into zero point 7 5. So, this would be 9 volts because of the drops in the switch these are not ideals switches and drops in the diode you will get slightly less than 9 volts. So, that is why in the plot (Refer Time: 25:53) will get around between 8 and 9 volts. We could also have look at the current plot a l. You can see here what to plot you have the (Refer Time: 26:12) the moment anything hash branch is there means their current signals available for you to see and these are the node voltages.

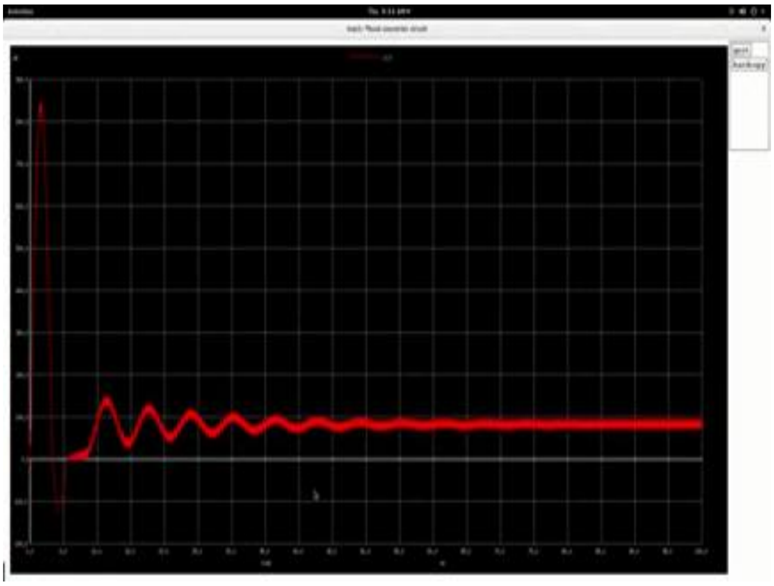
(Refer Slide Time: 26:22)



With the current signals these are all the transients and steady state occurs somewhere here. You can just take two cycles. So you see the inductor current rising doing the one time magnetically it is getting charged and the inductor current is falling during the off time this is when (Refer Time: 26:56) getting this charged and gets pumped of the output. So, the simulation works and you can see the various waveforms to understand the buck convertor circuit. We shall now go back to our notepad and the study a bit more on the waveforms of the primarily (Refer Time: 27:21). You should play around with these because we have just put arbitrary values. You can improve the dynamics by reducing the capacitance value and also play around with the l value to adjust the ripple current value and then save it and you can try what happens.

So, go back to this form. You use the buck arrow to get back to that command line generate the at least again then go into n g spice run plot. So, let us say once you do that.

(Refer Slide Time: 28:14)



So, you see the current dynamics are improved; it has reached the steady state earlier when in the earlier case when capacitance was 1000 micro currents. So, in this way you can try to learn with more about circuit by changing the parameters and now allowed in about the currents and the voltages of the various critical (Refer Time: 28:38) and branches. Then it closes it quit n g spice and that completes the simulation.

In the case of boost and buck boost converter what you to do is change the position of the switches diode and the inductor according to their respective topologies and we do the net list and then run the simulation. I (Refer Time: 28:08) you for trying out the boost in the buck boost converter along same lines as we did for the buck converter.