Analog Circuits and Systems through SPICE Simulation
Prof. Mrigank Sharad
Department of Electronics and Electrical Communication Engineering
Indian Institute of Technology, Kharagpur

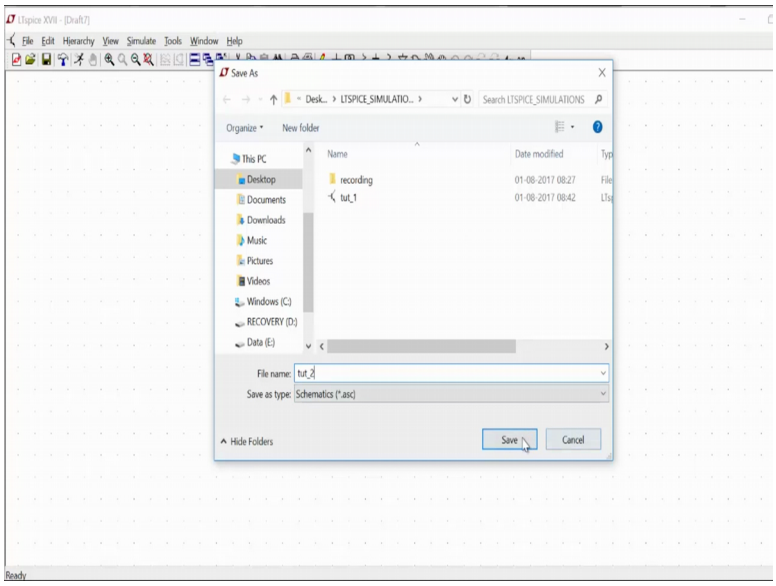
Lecture - 17
Tutorial II

Hello everyone. Myself Ankit Shivhare, and we are in the session 2 or it spice simulations. So, earlier we have done tutorial one session where we have discussed on the voltage to alter how to simulate we have done DC operating for simulations right and we have seen a net list, we also seen how to set up the background and all these things right, cursor with complete data. So, in the second part we are going to simulate a voltage control voltage source.

So, what does voltage control voltage source is suppose we are applying a certain input voltage the output voltage will be depended on that input voltage? So, suppose you are applying suppose 2 volt input source and suppose your voltage control (Refer Time: 00:54) was gain is 2. So, the output will be 4 volts. So, it is a multiplication in a way right. So, you are having some gain value and you are applying some input.

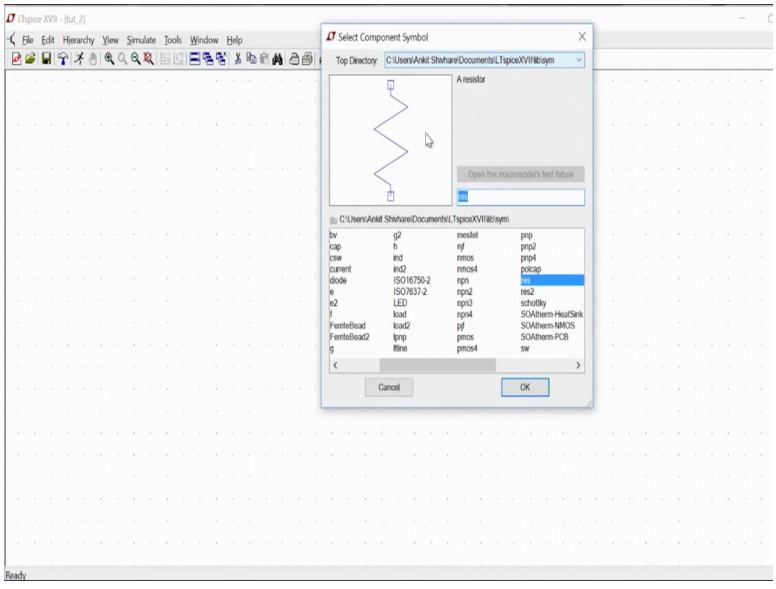
So, where this is used is like suppose (Refer Time: 01:04) there whose gain is 10 to the power of 6. So, we are applying some main macro volt. So, we will get the output in some volts right. So, it makes some sense right. So, we have a some input which is of very low amplitude it needs to amplify. So, in a way it is a voltage control voltage source. So, we will do simple simulation of voltage control voltage source we will also do op amp up and all these simulations later on. So, all these article session understanding that we are trying to do right.

(Refer Slide Time: 01:34)



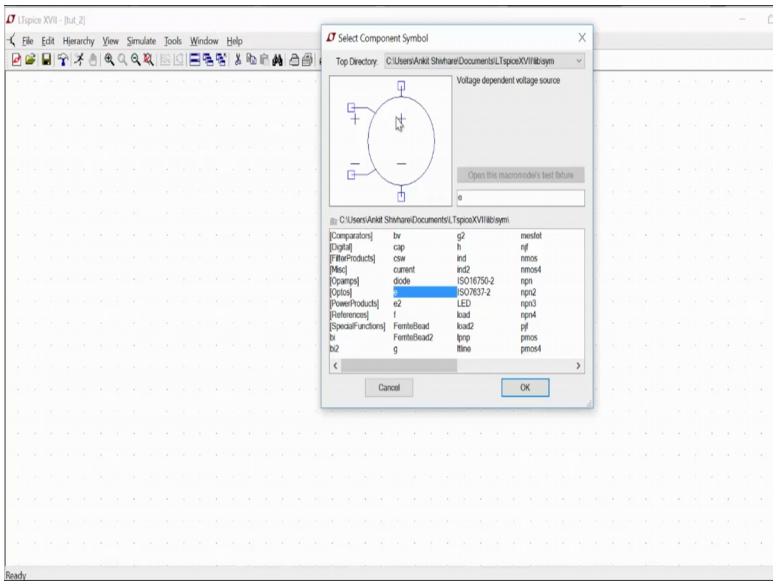
So, coming to the session; so, first we will do the new schematic I have done already now I should save that schematic. So, first is the save as option. So, now, this tutorial session will be named as tutorial underscore 2 and we should save it. So, it will be in dot asc format.

(Refer Slide Time: 01:58)



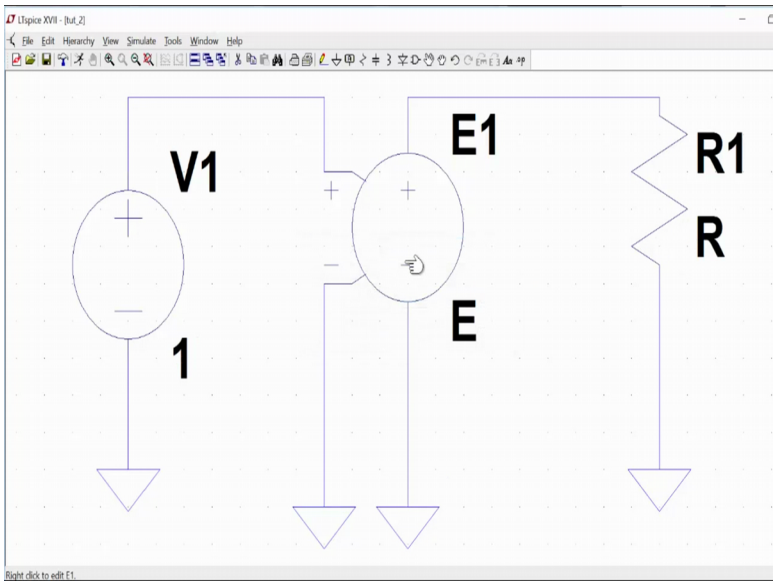
Now, for the simulation we need a voltage control voltage source you just go to the component and you go to the E E and E 2.

(Refer Slide Time: 02:07)



So, you will see. So, you suppose you see this is the voltage control voltage source now there is one more that E 2 it just the swap. So, like negative positive right. So, the input is a swap.

(Refer Slide Time: 02:26)



So, it is the same thing, but with the different means differentiations right different things format. So, now, you can have this voltage control and you can keep it here.

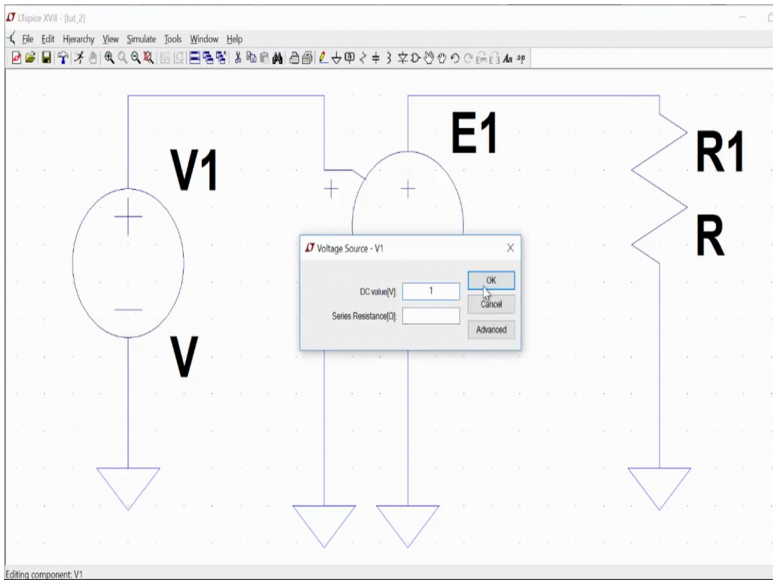
Now, this is the input set we need a voltage source. So, let us have a voltage source. So, DC voltage sources the normal voltage source that is voltage and we will go and we will

place it here and we will need a resister a resister res. So, resisters we will need PQR, right this is the letter re. So, a resister is here. So, we can do control r and next and control name is system mirror. So, we will do does not make any sense here.

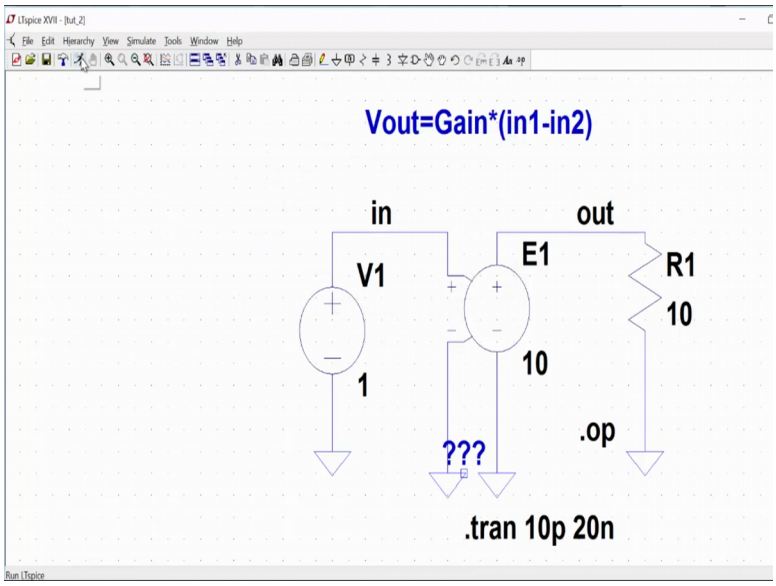
So, when you are placing it actually when then you have do next go directly here and control R. So, I need R 10 kilo ohms suppose resister. So, multiple times I have done. So, its and we need to have a ground right. So, ground we can take one common ground or we can take three separate ground totally it does not make much difference in the net list that will not make any much difference here.

We should connect all these appropriately. So, we need a ground for this little drag this one and we need a more ground. So this, drag and move option are very best actually and it is very comfortable actually the selt is spice that is why we would the selt is spice. So, it is space bar you can fit your screen sometimes it may move. So, just connect it back lot of connections are there you can make the all ground same it does not matter.

(Refer Slide Time: 05:12)

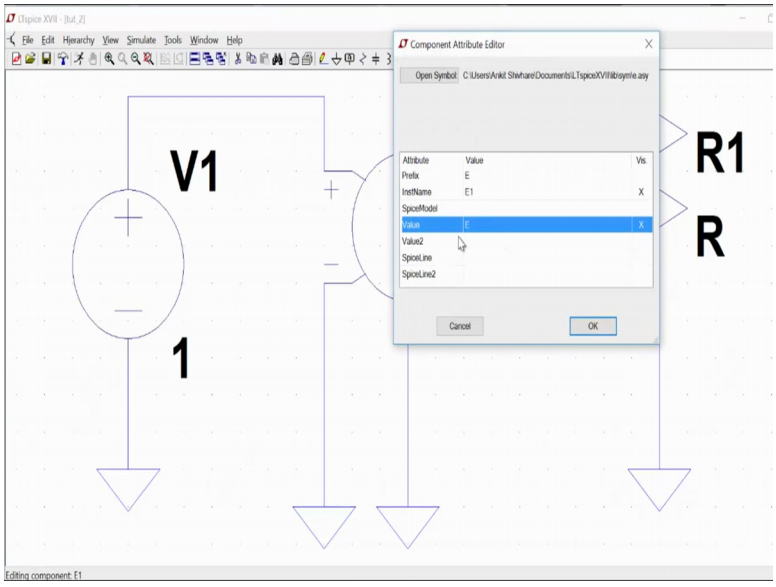


(Refer Slide Time: 05:16)



So, now I need to set the receiver to the (Refer Time: 05:10). So, right click on this and you will get option DC value I am saying it is suppose one word and I need to set the gain of this program. So, it is asking me. So, what is the value one value 2 the instance name?

(Refer Slide Time: 05:21)

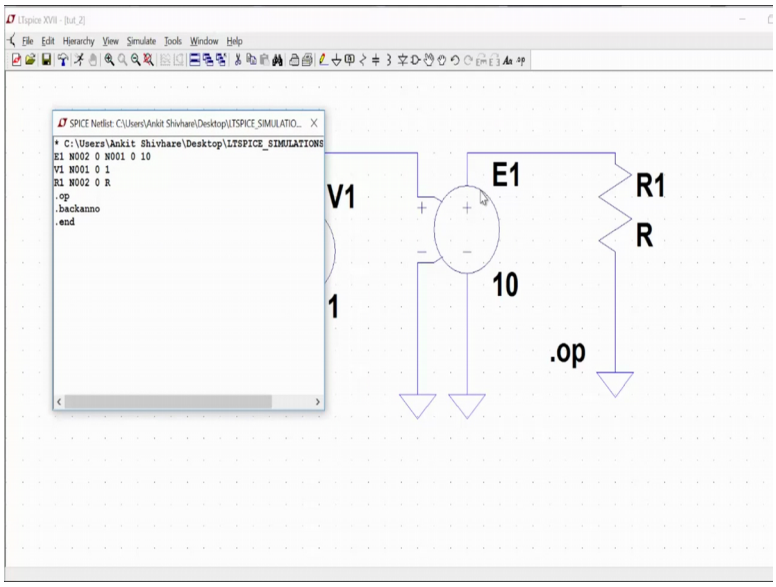


So, you can see this is the prefix that is E and this is the instance name. So, suppose I got second voltage control voltage source the instance name will be change E 2, but the

prefix will remain same right. So, it is a E and the value of this I can set this suppose I am saying I want the gain of 10. So, I am getting gain of 10 from here.

Now, I need to do simulation. So, I need to set up the DC operating point. So, I can say this (Refer Time: 05:49) dot op and just plate it here that is it and you just move this way now I showed see the net list first right. So, view and I can go to the spice net list outlook like.

(Refer Slide Time: 06:12)

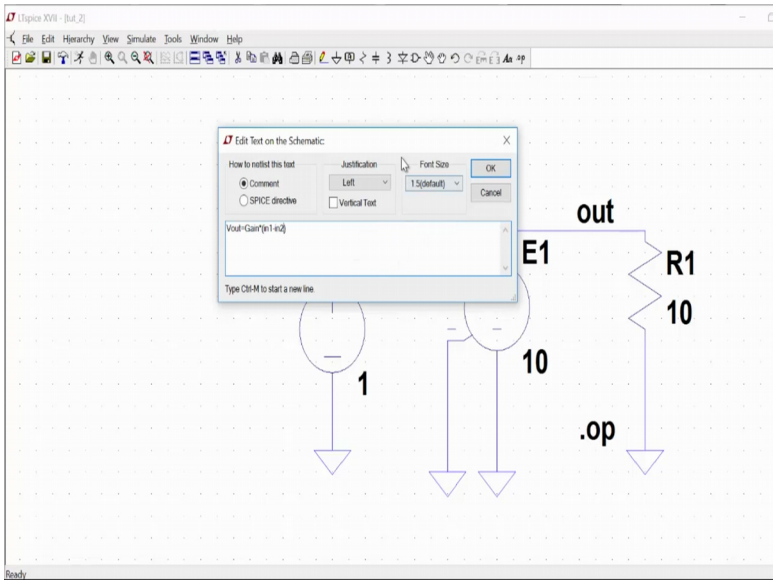


So, you can see E one is there with N 0 0 2 this is the net and 0 and 0 0 1 0 10. So, this is different 4 net alternate rates. So, 4 nets between N 0 0 2 1 is ground and N 0 0 1 and then 0 and the value of the gain is 10 right when v one voltage source again this N 0 0 1 2 nets 0 and one the value is one r one same with resistance is r I am not given that r value. So, I should keep that r value something right make some sense. So, now, this is done.

Now, I can label this net because I seen N 0 0 1 N 0 2 because that difficult for me right. So, I can label them that like this is my input in. So, I can just place it here and I can place another label stating that this is my output right this is my output. So, now, I suppose I see the net list (Refer Time: 07:16) and net list see here now. So, this is out in. So, v one is between input and 0 and out 0. So, it makes some sense, right.

So, these nets will define a net list with the proper way. So, if you do no;t if the spice do not need always this GI window you can just going to command terminal and you understand.

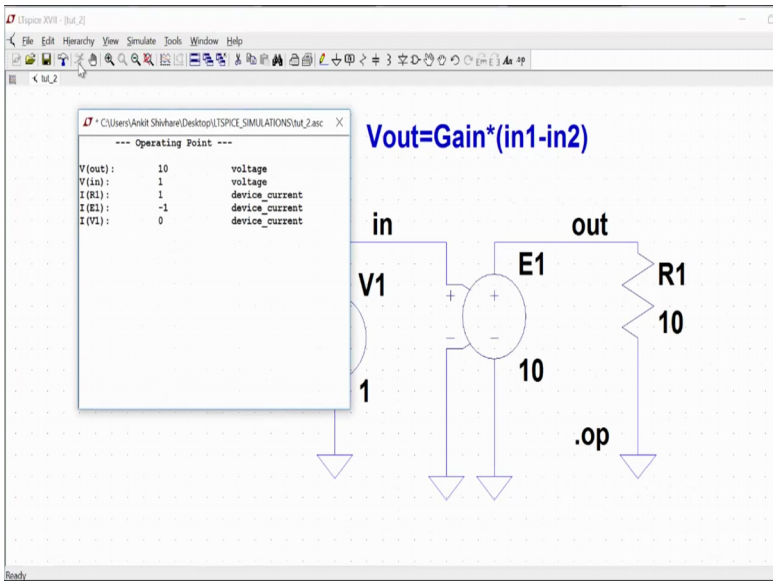
(Refer Slide Time: 07:45)



And you can just directly type and do the simulation right it make some sense and I can add also comment also. So, like I want to write like what is v out here is nothing, but gain that is gain times.

So, gain times in one minus in 2. So, I can write this comment I in one is nothing, but this input and this input. So, the difference between these 2 inputs multiplied by the gain that is 10 here will give the output that is how these were comment only it is not any syntax or something just this is the comment I have given you can delete that comment no matter for understanding I have just written that.

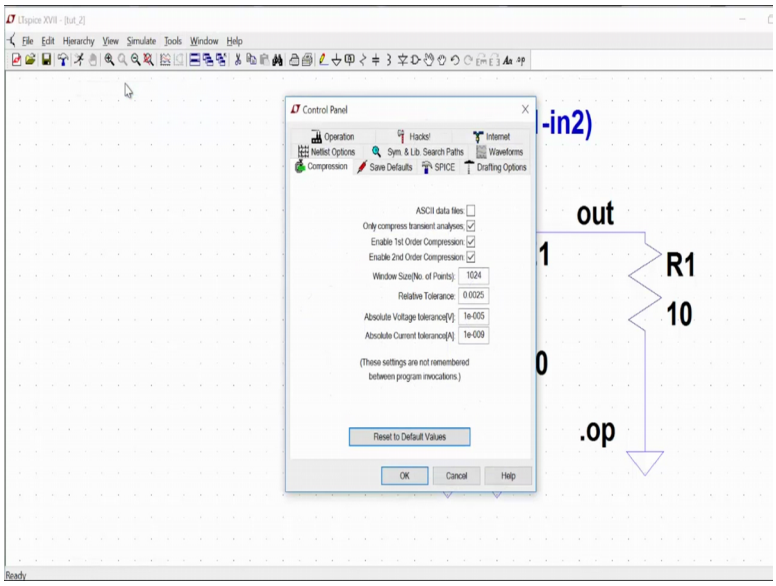
(Refer Slide Time: 08:31)



And I can do the round just see what goes up now you can see the output that is 10 and input is 1 I R 1 is 1 I E 1 is minus 1 and I 1 is 1. So, since the resistance the voltage here is 10. So, the current that will flow through the resistance is what one ampere correct the current in the voltage source this will be negative right because the current is in the voltage source and you see the current is going from the negative to the positive and here it is going from positive to negative.

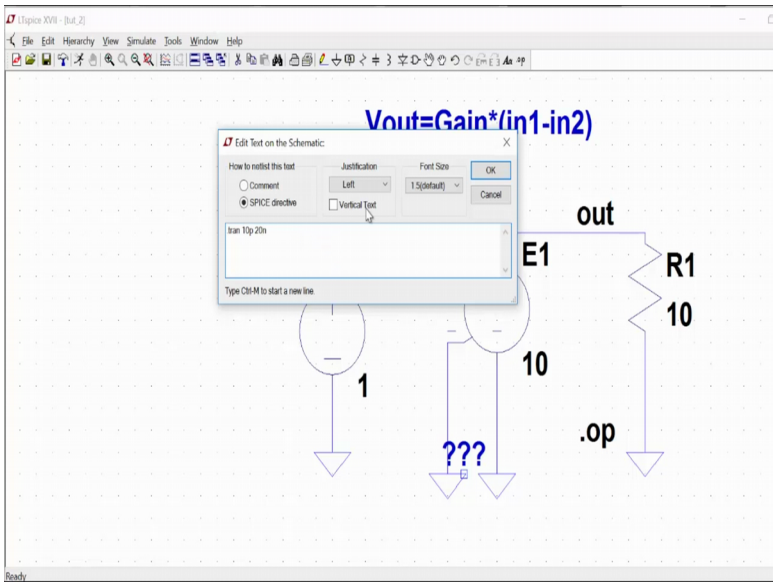
So, current flows always from positive to negative in the device right. So, is going negative to positive. So, current if negative here positive to negative. So, current is positive and I E 1 is minus 1 and I V 1, because no current is being taken up right voltage when the voltage source means what the current input current is 0. So, the input resistance here is what infinity. So, no current is there it is a ideal voltage source. So, this is making some sense.

(Refer Slide Time: 09:41)



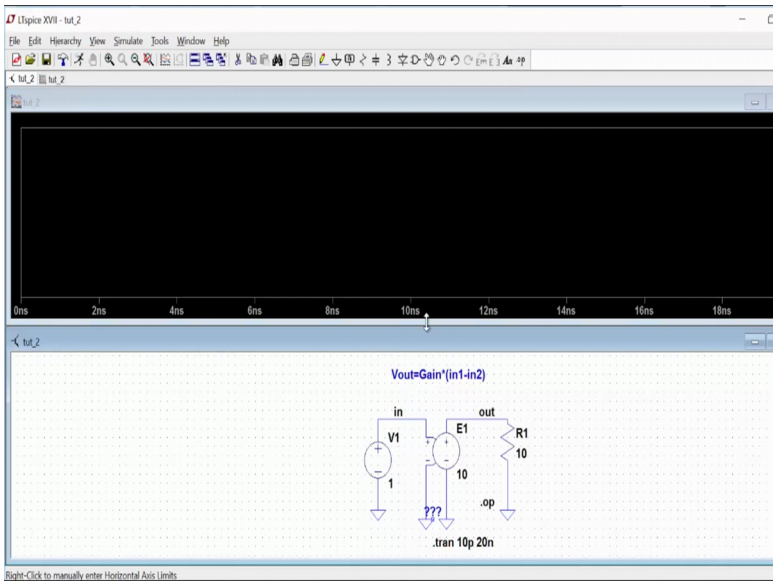
So, voltage control voltage source is done we can also do the same thing in the transient picture also. So, we can have the transient simulation also for this case. So, let us see how the transient simulation will look like. So, I can go to control panel. So, I need to set up the transient times simulation for this case. So, for this I can go to view and edit spice directive. So, now spice directive come.

(Refer Slide Time: 10:12)



So, I want to go for a transient simulation. So, I will write dot try then the simulation (Refer Time: 10:22) is 10 Pico then I will go to 20 nano.

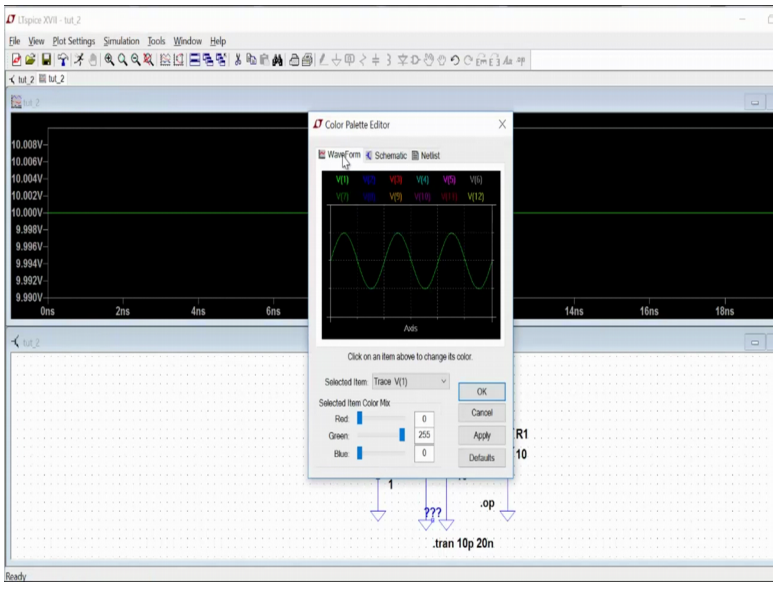
(Refer Slide Time: 10:32)



So, this is and I can place it here and can make around and let see away from GI window we open I can see; what is the voltage I want to measure right. So, I want to see the output. So, output is 10 volts now I can set the view from here also 2 different colors right.

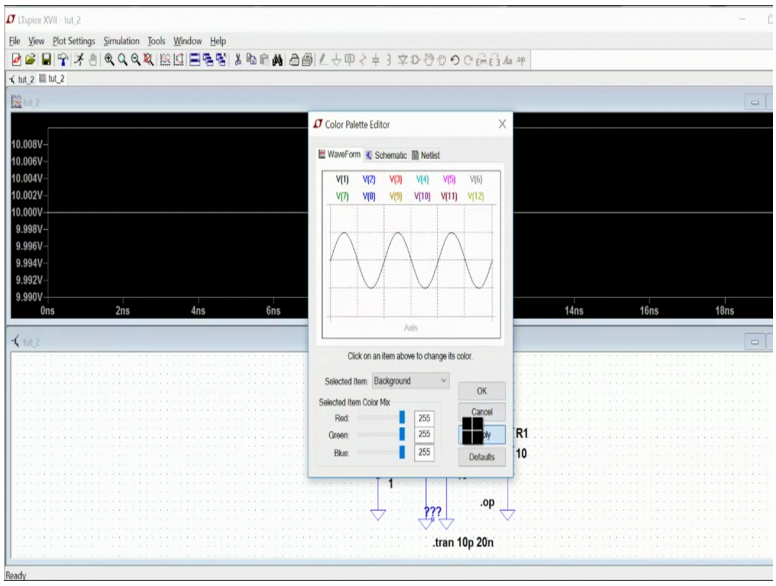
So, I do not want these colors. So, it is asking me simulation tools colors preference.

(Refer Slide Time: 11:07)



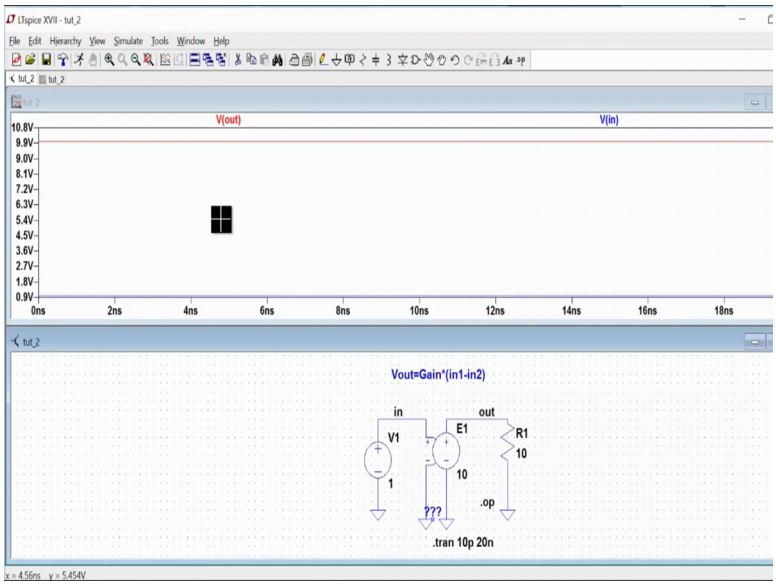
And wave forms I can say I want a light background sorry this is for this one I want for background this to be light white.

(Refer Slide Time: 11:22)



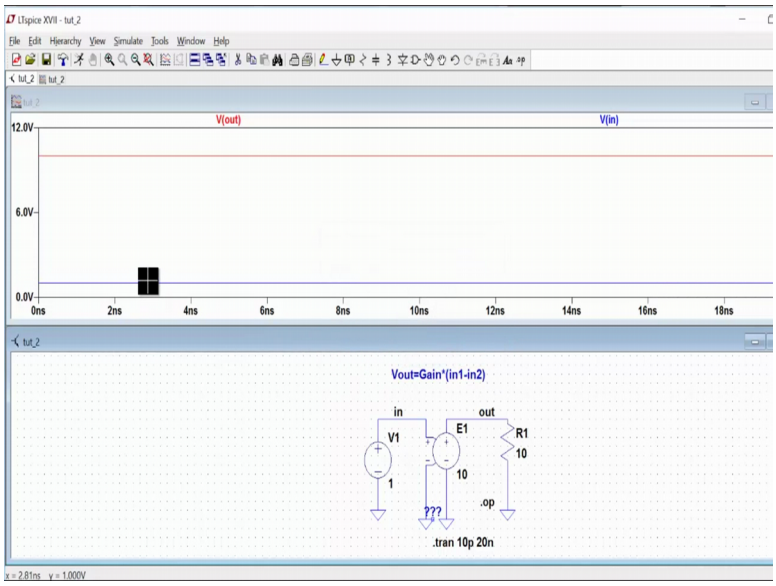
So, I can do that apply and I want to axis to be perfectly black apply and axis is black and I want to do see the traces with some red color. So, it is making some scattered; so, 10 volts.

(Refer Slide Time: 11:38)



Now, I can also do the voltages here input. So, this is one volt right. So, this is one volt. So, this is one volt here. So, I can just see the axis the axis I can make it 12 volt and bottom axis is 0 volt. And now I can do?

(Refer Slide Time: 12:03)



So, this is 10 volts and this is one volt the input signal. So, I can also measure the current that is going to be there resistance r one. So, in this axis you can see the current that is going is 10 amperes. So, 1.0001 ampere because 10 volts right; so 10; 10 ampere. So, this is 1 ampere that is current you can (Refer Time: 12:26) right this one. So, this is the transient picture because everything is DC here. So, means I am seeing all that DC value for all the transient time. So, the transient time is in the axis that is 0 to 29 nano seconds.

So, hope this make some sense we will do further discussion form this in the next tutorial session.

Thanks a lot.