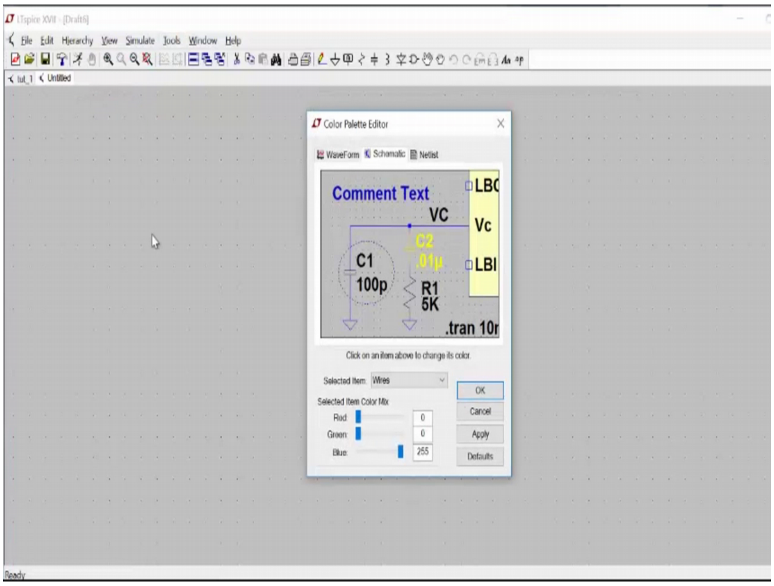
Analog Circuits and Systems through SPICE Simulation
Prof. Mrigank Sharad
Department of Electronics and Electrical Communication Engineering
Indian Institute of Technology, Kharagpur

Lecture - 16
Tutorial I

Hello everyone. Myself is Ankit Shivare, today we are going to have a session of lattice spice. So, this is our first tutorial.

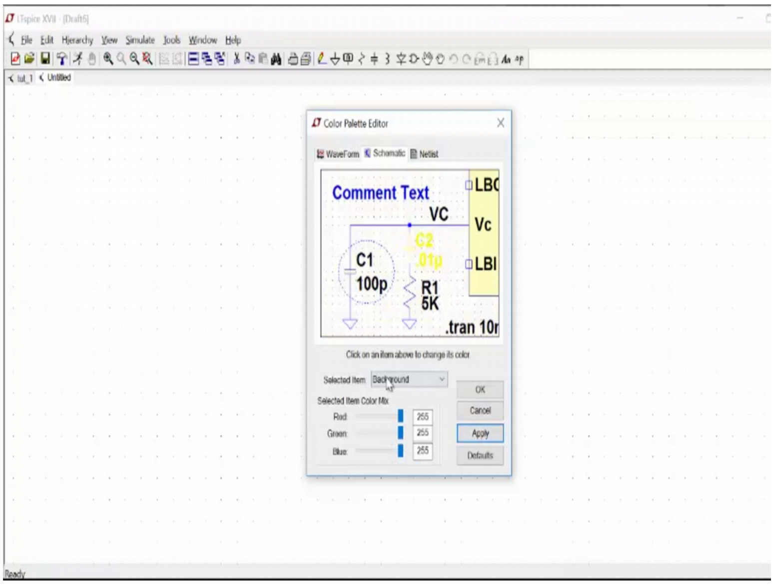
In this tutorial we are going to see the different options that are available lattice spice, the tools control panel options then we are going to see; what are the all different windows that are available in lattice spice. Then we will go to the tutorial first we are going to think on voltage divider. So, let us have a voltage divider circuit. And see the wave forms how it looks, then how to setup the different window planes etcetera, that also we will see ok.

(Refer Slide Time: 01:44)



So, first we need to install LTspice XVII as per the tutorial, as per the link I have sent you ok. You try to install it you try to install if you find any issue then revert back.

(Refer Slide Time: 01:49)

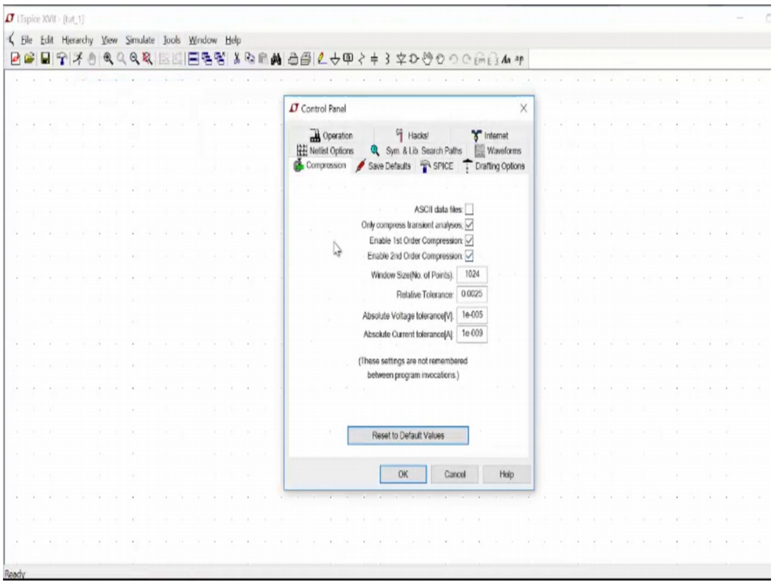


So, what are the different options that are there? Files you can see there is no schematic. So, you can see you can have a new schematic. So, window like this will open up right. Then we can have save options etcetera. Then this is spice directory capacitor inductor diode and all these things we will get. So, we will see how these all these things use there is new symbol view there are spice network option and all these things.

So, first and important thing is the colour preference how you want to set up the colour preferences. So, like for background suppose I want a white background. So how I will do that? So, you can see when I do this though this is now all right white background right, it is looks better for me. So, I will set a white background for this. So, for all the other things I can satisfy I can, for grid I can do this. So, this will be a black dots that will be coming, probably in the video you may not able to see, but this is the black dots that will come up.

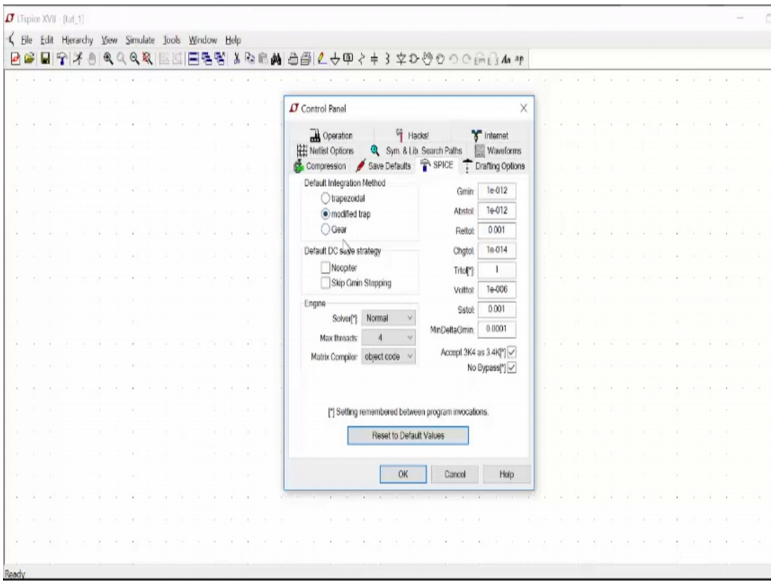
Now, first we need to save in LTspice simulation. So, first tutorial I have saved in tutorial 1. So, I can save it lattice tutorial 1. So, do you want to replace here yes I set and I can close this one. So, this is how I can save the tutorials.

(Refer Slide Time: 02:21)



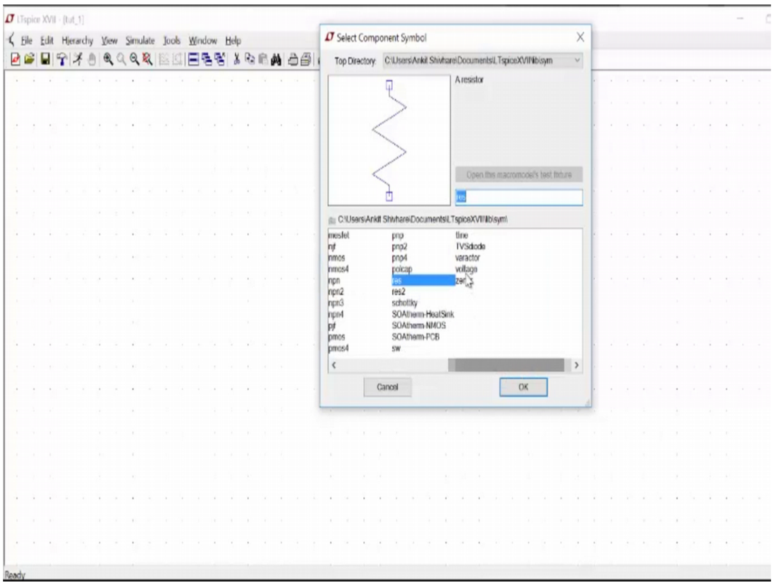
Now in tools control panels there are lot of options colour preference, I told you colour preferences control panels you can see that, these are the compression that will occur. Absolute a voltage tolerance (Refer Time: 02:31) this will have a discussion on this. It is means how much error you can tolerate actually in a simulations. So, you cannot take it tutorial also, you cannot take it very low value also it is; however, you have to see that width solution is good.

(Refer Slide Time: 02:49)



So, these are integration methods. So, that spice simulation will occur by these numeral integration methods. So, these are trapezoidal modified trap gear 3 methods are available. There are more options available with the higher with the cadence spectre lot options are available right. These are gmin, absolute relative tolerance Chg tolerance, Tr tolerance all these tolerance you we will discuss later. So, let us start with this and we will have a discussion on this further. So, first we need to draw voltage rider.

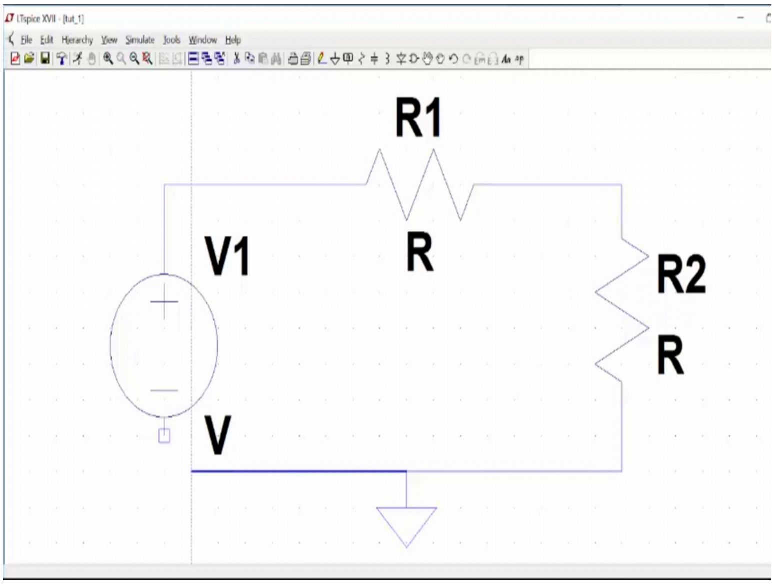
(Refer Slide Time: 03:28)



So, for that we need to resistances and a voltage source correct. So, let us have voltage source we have go to the component section and you go to the voltage option. So, you place here then you need 2 resistances as well. So, you need a register. So, you can have these. So, now, you can have 2 keys that is control R and control K, control R is for rotation and control K is for mirror.

So, control R we can we can do like. So, this is the 2 resistances we can draw. So, these are the 2 resistance have draw. So, always we need a ground for all these things. So, we can have this ground and connect here. So, spouse bar is split a screen, now we can draw the wires and connect all the components. So, a close loop is formed.

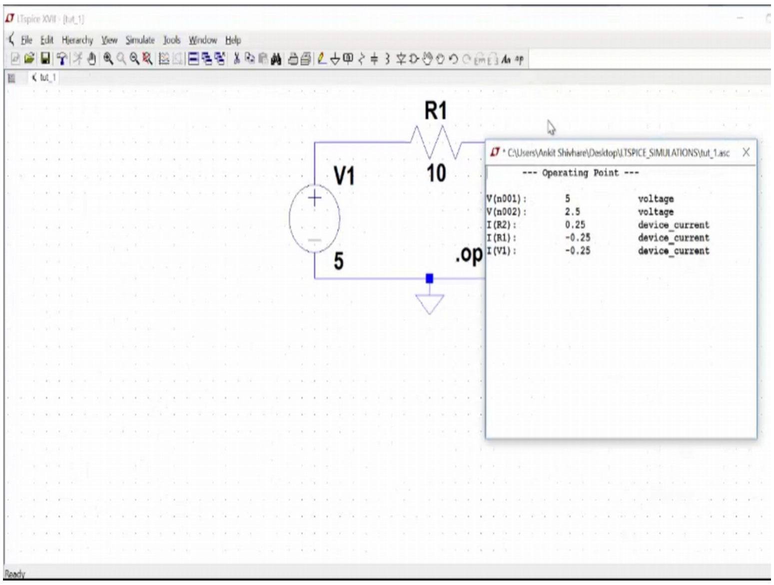
(Refer Slide Time: 04:27)



Now, we need to set the values of these voltage sources. So, let us see what will happen. So, right click on this and you will get these options like DC value. So, let us, let us assume that we are putting a 5 volt DC value and what are series resistances. So, let us say we are not putting anything here right. So ok I will set. So, this is the V1 and this is the 5. Similarly we can set the resistance value. So, tolerance is there power rating is there like how much power you want to have. So, let us now setup a simple timings and timings 2 resistances. And this is how your circuit will look finally.

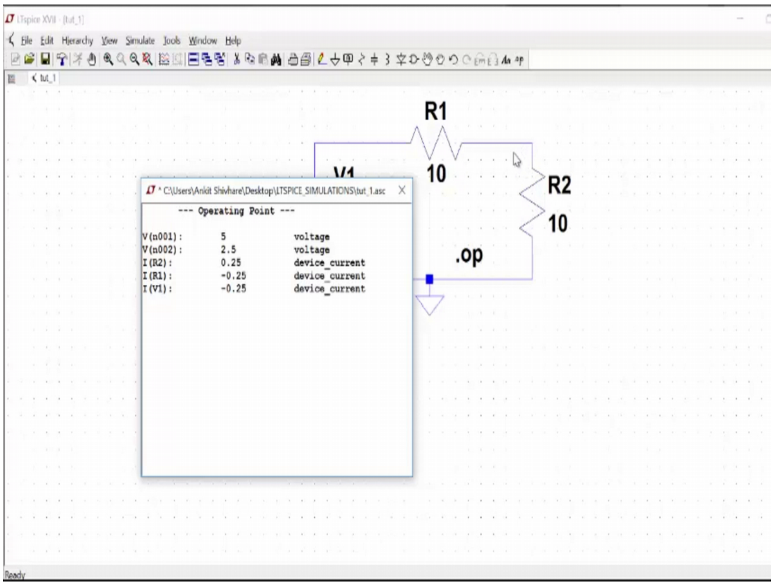
Now, we need to do the simulation of that. So, we need to set up the operating point. So, spice directory recall. So, we can said that we have we need to have a DC operating point. So, we put dot op and you just place it here in this window. So, the spice will understand now here that operating points needs to committed for op, ok.

(Refer Slide Time: 05:49)



Now, we need to simulate the simulation. So, when we run that, So see the window will pop up. So, these are the different net that we show. So, like this V n 001. So, we do not no exactly that is V n 001, but we can think like this is the voltage source this has to be V n 001 right. So, this is 5 volts. So, it is naming this net as n 001. And this is n 002 that is 2.5 volt because, 10 ohms to 10 ohms voltage is fitted.

(Refer Slide Time: 06:10)

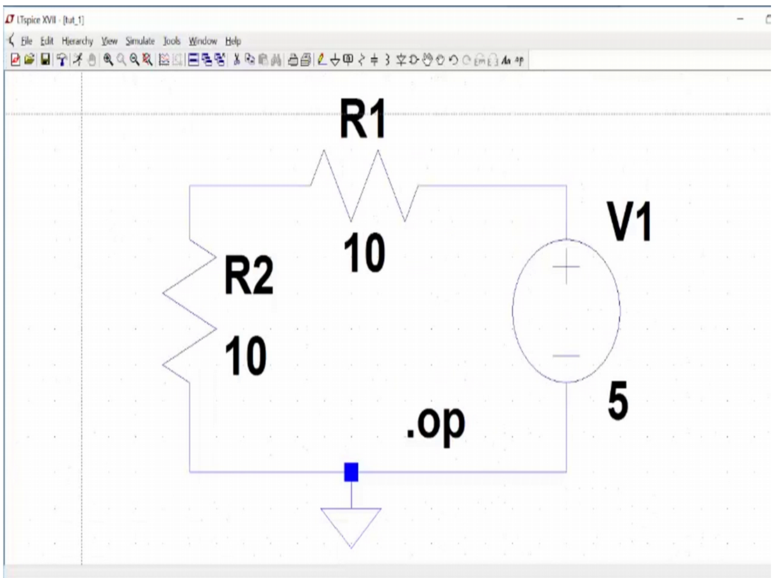


So, it will be 2.5 volt. So, this is voltage n 002. And these are the currents because you can see in the window options I have save the all the device currents. So, it is saving all the currents; so IR 2 that is 0.25 ampere. So, 5 divided by 20 ohms. So, that is what 0.25 right. So, this much current is flowing.

Now, you can seen that current has shown negative here as well. So, what does this signify is the current always it seems from the positive to the negative. So, means current is always from the positive to the negative in spice simulations. So, here current is going from the positive to negative right. So, inside this current is going from positive to negative, but it should be right this way this is positive this is negative. So, it should be awaited. So, that is why it will positive. So, suppose I place voltage source here source here and the resistance here than the current will be just opposite. So, we can do that thing let us do that.

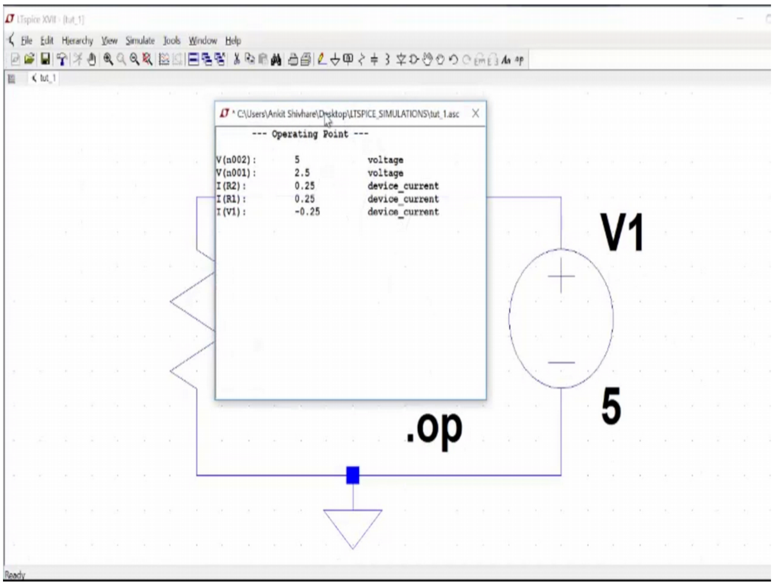
So, I just correct up this net, this net, this net and just simply. So, we can move that. So, this is the move option. You can just place it here, you can move this voltage source to here. So, it is little bit fast let it be here and we need to again cut it, cut these nets, floating nets and connect back wires. So, let us see what happens.

(Refer Slide Time: 07:56)



So, let us go to the space bar fit and now and now go to the spice netlist. We can see the spice netlist, this is the spice netlist. So, this is V1 voltage source between 2 nets and 002 and 0 that is ground and that are voltage is 5 volts. R1 resistance between nets and 002 and 001 the value is 10 ohms. Similarly R 2 resistance and 001 and 0 the value is 10 ohms we are collecting the operating point without op and always the spice netlist ends with dot end ok.

(Refer Slide Time: 08:40)



So, now again we can run the simulation. Now you see the current has changed to positive in the resistance values. Because the current it is asking from the positive to negative side rate. So, this was for now the current is going in this direction. So, hope it make sense right. And the voltage of current through the voltage source is negative now.

So, this is how it goes. We will have next session and there we will see voltage control voltage source.

Thank you.