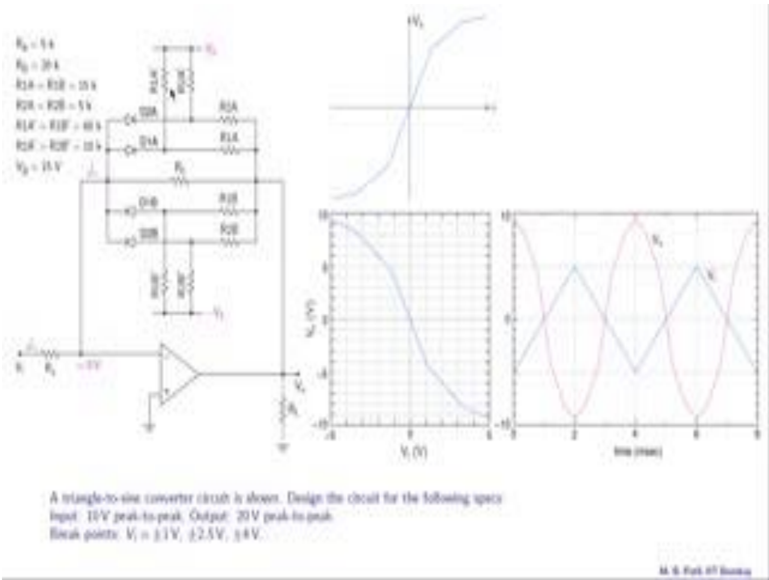
Basic Electronics
Prof. Mahesh Patil
Department of Electrical Engineering
Indian Institute of Technology, Bombay

Lecture – 49
Simulation of triangle-to-sine converter

Welcome back to Basic Electronics. In the previous class we looked at an op-amp circuit where conversion of a triangular input voltage to sinusoidal output voltage. We will now take up a design problem based on that circuit first we will find the resistance values required for the given specification after that we will simulate the circuit from scratch which includes entering the circuit schematic setting parameter values and simulation related parameters.

Finally, we will run the simulation and verify that the output voltage is indeed what we would expect. We will also look at the Fourier spectrum of the output voltage to judge the quality of the output wave form, let us begin.

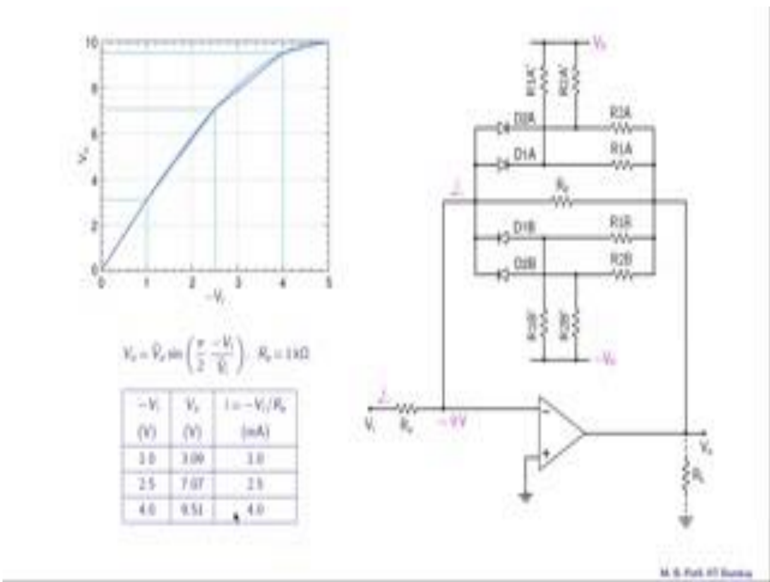
(Refer Slide Time: 01:11)



Let us consider this following design problem a triangle-to-sine converter circuit is shown this circuit here which we have seen earlier. Design the circuit for the following specs, so the same circuit we need two design for a different set of specs. Input voltage ten volts peak to peak output 20 volts peak to peak and the input of course, is triangular and the output is a sinusoidal and we want the following break points in V i; that means, along this axis plus minus 1 volt plus minus 2.5 volts and plus minus 4 volts.

Let us briefly go over the operation of this circuit what we did first is to plot V o; that means, this voltage as a function of this current i and that end out to have this shape here and then we figured that in this op-amp circuit V i and i are related by V i equal to minus R a times i and because of the minus sign the V o versus i relationship got flipped and we have V o versus V i as shown in the figure here and these are the time domain plots that is our input voltage and that is our output voltage. So, now, we want to design this same circuit with three breaks. So, therefore, we are going to have three branches like that, we have two here we will have one more branch and of course, we need to find out all the component values this resistance, that resistance, that resistance and so on.

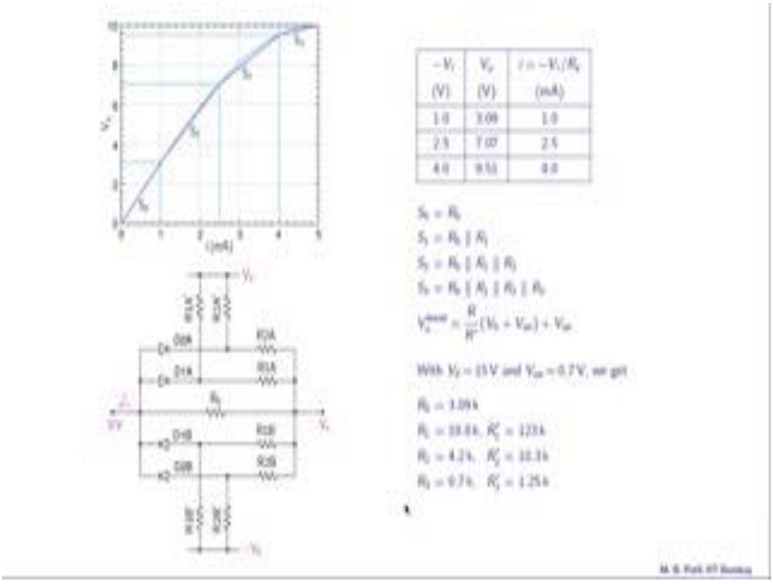
(Refer Slide Time: 02:52)



So, this is what our V o versus V i should look like and note that we are plotting V o versus minus V i here because we know that V o is proportional to minus V i and not plus V i. And this expected relationship, the sinusoidal relationship here shown in the light blue colour is represented by this equation here V o equal to V o cap that is the amplitude of the output voltage which is 10 volts times sine pi by 2 times minus V i divided by V i cap and V i cap is the amplitude of the input voltage in this case 5 volts.

Since we know the break points in the input voltage V i we can calculate the corresponding break points in the output voltage V o with this equation now and those values those break points in V o as shown in this column here. For example, for V i equal to plus minus 1 volt we would have V o equal to plus minus 3.09 volts and so on. Also let us take R a equal to 1 k and with this value since i equal to minus V i by R a the numerical values of V i and i would be the same except for the minus sign because R a is 1 and therefore, V i equal to 1 volt would mean i equal to 1 milliamp and so on. So, now, we have the complete information about the break points we know the V o value for each break point and the current value for each break point.

(Refer Slide Time: 04:40)



Let us now look at the component values that we need to calculate and as we have seen earlier because of symmetry R1A and R1B are equal also R1A prime and R1B prime are equal. So, the values that we need to calculate are R 0, R 1, R 1 prime, R 2, R 2 prime and there would be a third branch because we have three break points. So, we will also need to get R 3 and R 3 prime and we will assume that these diodes turn on at 0.7 volts we will take this V 0 as 15 volts. So, this minus V 0 would be minus 15 volts and now let us see what information we already have. We have these break points in the input voltage and we have calculated the corresponding break points in V o as well as in the current file.

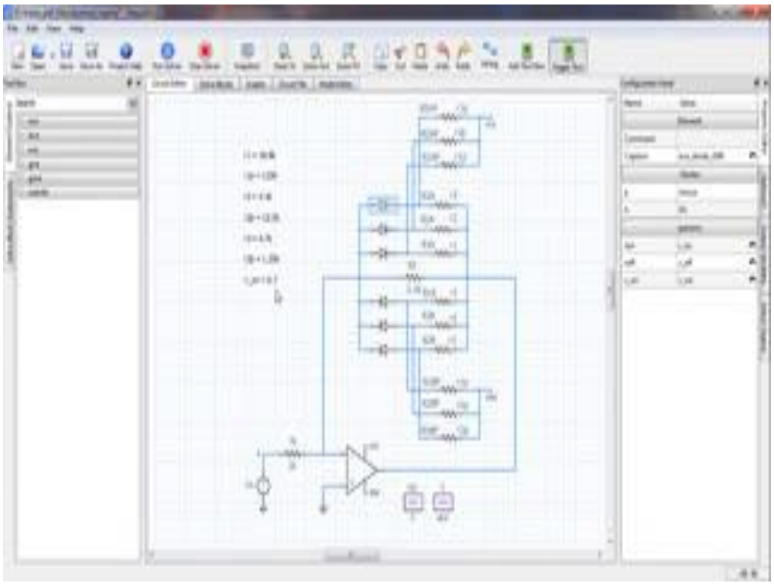
Here is the plot of V o versus i and these are the break points one two three in addition of course, we know that with i equal to 0 we will have V o equal to 0 and with i equal to 5 milliamps we have V i equal to minus 5 milliamps and that corresponds to the maximum value of the output voltage that is 10 volts. So, we have these 5 data points now what we will do now is to connect the first two with a straight line, the next two with a straight line and so on and calculate the slopes S 0, S 1, S 2 and S 3. As we have seen earlier these slopes can be related to the resistance values as follows the slope S 0 is equal to R 0, S 1 is R 0 parallel R 1, R 0 parallel R 1, S 2 is R 0 parallel R 1 parallel R 2 R 0 parallel R 1 parallel R 2 and so on.

And since we already have completed these slopes now we can find the resistance values R 0 is the same as S 0 R 1 can be found using S 1 and R 0 R 2 can be found from S 2 R 0 and R 1 and so on. So, at the end of this step we have all these resistance values namely R 0, R 1, R 2 and R 3. The last step is to calculate these resistances R 1 prime R 2 prime and R 3 prime for the third branch which is not shown over here and as we have seen earlier these are related to the break points in the output voltage namely this break point this break point and this break point. For example, this first break point is given by R 1 by R 1 prime that is R 1 by R 1 prime times V 0 which we have taken as 15 volts earlier plus V on which is 0.7 volts for the diodes plus V on. Now we already have R 1 and knowing this break point in V o this value here you can now calculate R 1 prime and similarly we can get R 2 prime and R 3 prime.

So, here are our final results with V 0 is equal to 15 volts and V on equal to 0.7 volts we get the following values for R 0, R 1, R 2, R 3 and R 1 prime R 2 prime R 3 prime and you should definitely go through this exercise of calculating this values and make sure that you end of with the same values.

You might notice that there is some reputation involved in these calculations and you might want to write a small program to perform these calculations it might help. So, now, our design is complete and let us simulate the circuit and make sure that it does not it work as triangle-to-sine convertor as we want.

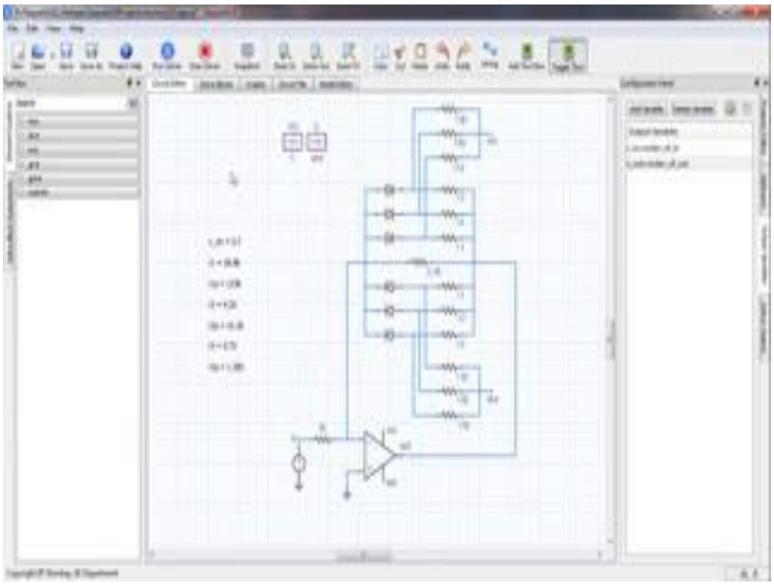
(Refer Slide Time: 09:09)



This is what our final circuits schematic is going to look like and we are going to construct it from scratch step by step so that you can also follow and make it up yourself. But before you begin let us note one specific feature that is very useful in this specific case and that is we have this global parameters here for example, V on has been specified as point seven volts.

Now there is a reason for that all these 6 diodes are going to be identical and they have parameters like on resistance, off resistance and V on, and these parameters are going to be the same for all these 6 diodes and therefore, it make sense to have one global parameter called V on. So, suppose we want to change it from 0.7 to 0.6 then we need to only make one change over here. So, that is very convenient. So, let us remember this feature and let us now start constructing this circuit schematic.

(Refer Slide Time: 10:10)



Let us start with defining a few global parameters and that is done by right clicking on the canvas anywhere and choosing this add global parameters option and now we are asked for the name of the global parameter which is V on, let us add some others - R on and R off. So, we will use these for diode parameters. Now let us click anywhere in the canvas left click. So, that the parameters appear here and now we can edit this. So, V on can be 0.7, R on could be 0.1 and R off could be let say 10 mega ohms and when we bring in the diode now we can use these parameters to assign as properties of the diode. The diode model that we are looking for in this project is called diode r. So, let us bring that in like that and you see that this parameters are on resistance, off resistance and V on, it has got some default values what we will do is to assign the global parameters over here that we defined earlier.

Like that and we need 6 of these diodes. So, let us copy these and three of those need to be in the other direction. So, now, we have the diodes in place. Next let us get the resisters. The resister is called r dot ece. So, let us bring that in and we need 6 of these like that and now some of these resistances will have the same resistance values and therefore, what we will do is define additional global parameters for example, r 1, r 1 prime etcetera. So, here are although resistance values that we have defined r 1, r 1 prime r 2, r 2 prime and r 3, r 3 prime we have also assigned values to them and this is of course, as per the design that we just completed. Now let us assign these values. So, this resistance should get r 1 like that, this resistance should also get r 1 like that, we can display the property of this resistor and it shows r 1 since we assigned it already and so on.

Let us now start part of the wiring that we need to do let me move this component. Now for the wiring it turns out to be very convenient to use this element called connector - connector underscore e to indicate electrical connector and we can place it over there like that, it is a dummy element and it is used to only for wiring purposes it does not have any physical properties, all right. So, let me move these and place some additional connectors like that move this back. So, that is what we have got and we will do the same thing for this bank of resistors as well.

So, this is what we have got now we have also brought in this R 0 and assigned 3.1k to that in addition we have got this R 1 prime R 2 prime and R 3 prime as well on both sides up as well as down and now you can do the wiring and so on all right. So, this is what we have got after some of the wiring is completed and now we need to connect these three nodes together and that should go to vcc we also need to connect these three nodes together and that should go to vee. So, let us do that and now that we have placed one additional connector over here and we will name that node as vcc. So, let us click on this wire and call that vcc. So, we have got vcc here and vee here our next step is to make sure that these nodes are actually sitting at plus 15 and minus 15 volts and we will do that with DC voltage source and let us call vsrcdc box, it is called box because it looks like a box. We need two of these and we will make one of these the vcc source and the other one as vee source.

Now, this one let us say is our vcc source. So, its plus terminal would be vcc and the minus terminal would be 0 and we will make sure later that 0 is indeed our ground node. For the other source the plus terminal should be 0 and the minus terminal should be vee let us display the node names. So, all we need to do is to right click on the concerned node add node name text box and the text box comes over there indicating the name of the node we can move that with the arrow keys and if you want finer movement we can use the control and arrow key. So, let us do the same for the other nodes.

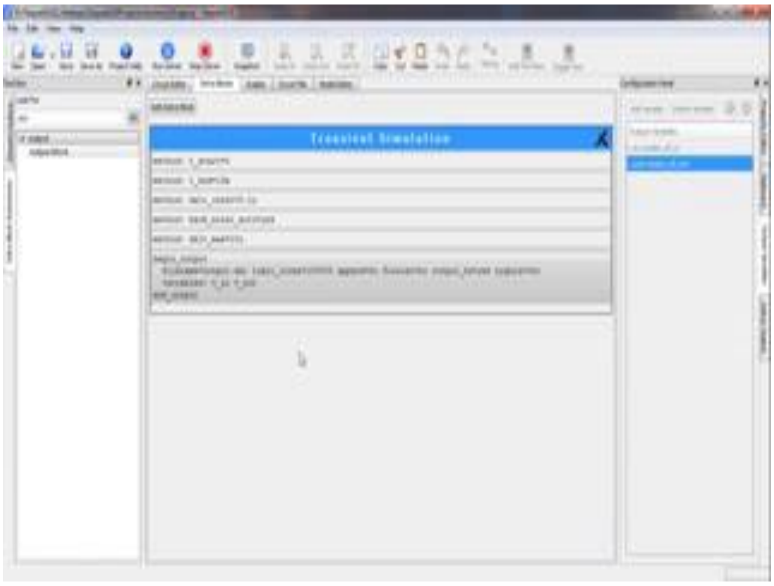
Next let us bring in the op-amp we will use the second floor one op-amp and in particular let us choose this 741 b. Now let us make connections with the rest of the network this minus terminal must be connected over here and the output terminal of the op-amp must go over there. So, this is what it look like we have also added this resistance 1k and we have named these vcc and vee nodes of the op-amp as vcc and vee; that means, this node is actually connected to this node because they share the same name and similarly the vee node of the op-amp which is named as vee is actually connected to vee, although we do not see a wire.

We need some more elements one is the triangle input source and that is triangle 3 here and the other one is ground. We can do some of the wiring. Now the other end of this source must be connected to ground we can either connect it like that or we can use ground dummy like that and let us remember to name this as 0 because we have used that name earlier and that indicates the reference node or the ground node.

Let us set the parameters for this triangle source now. So, click on it and frequency let us say 250 hertz the high level let us say 5 and low level could be minus 5. So, we have a 10 volts peak to peak triangle source with frequency 250 hertz.

The next step is to select output variables and we need the input voltage and also the output voltage we can name this as V in and this as V out. Note that we have named these nodes as in and out before we go to the solved blocks let us display these global parameter values because that would be handy for reference and the way to do that is right click in the canvas insert global parameter text boxes and then choose whichever parameters we like and then the list of the parameters along with their values will appear over there.

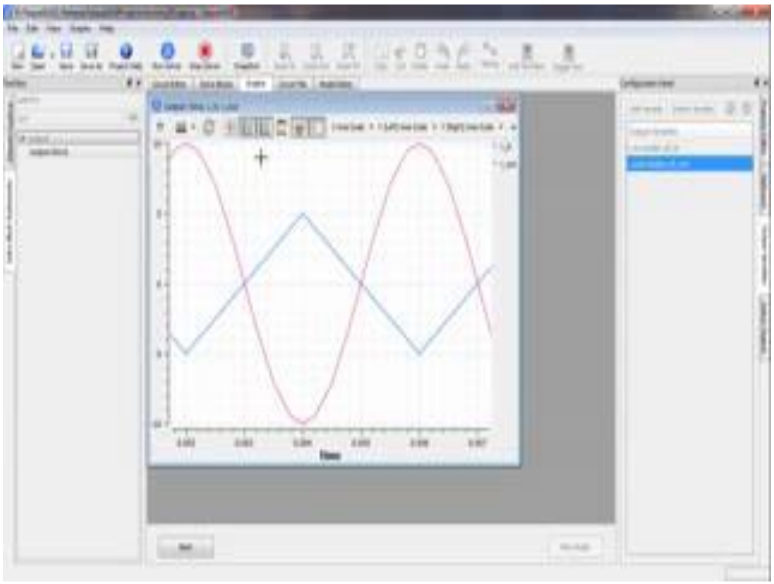
(Refer Slide Time: 19:59)



Let us now go to the solved box section adds a solved block, we want transient simulation, so make that transient simulation like that - our t end could be 10 milliseconds because our time period is 4 milliseconds corresponding to a frequency of 250 hertz. Our delta t constant, the simulation time step could be a small time step like 0.1 microseconds and in this case we are going to use the automatic backward Euler option. So, let us remove this and use the backward Euler automatic option.

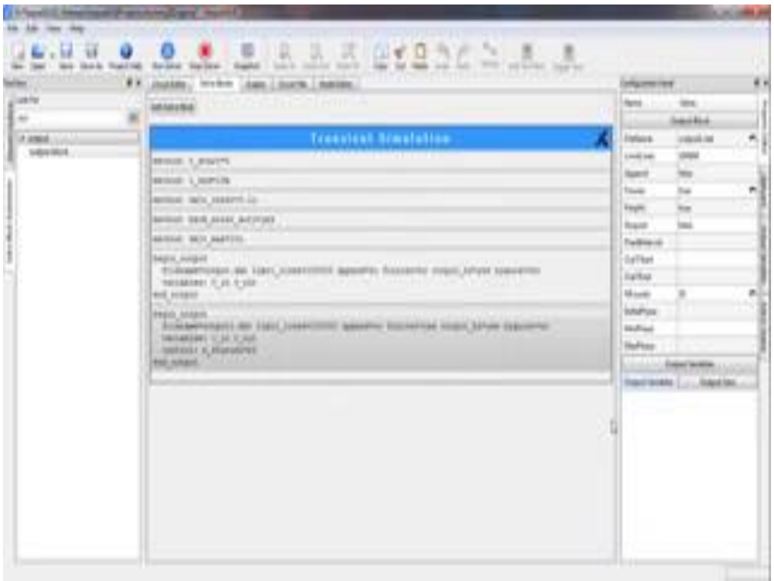
Let us also set a maximum time step of 10 microseconds like that and now what we need to do is to bring in the output block and let us add the variables that we would like to see namely input voltage and output voltage. Now we are ready to run the simulation, so we click on run solver the simulation is over; let us choose time as the x axis, and let us plot both of these together V in as well as V out and that is what we have got.

(Refer Slide Time: 21:24)



We can expand one period like that and that is our input triangular wave and that is our output sine wave. The input is 10 volts peak to peak and the output is 20 volts peak to peak as we expected.

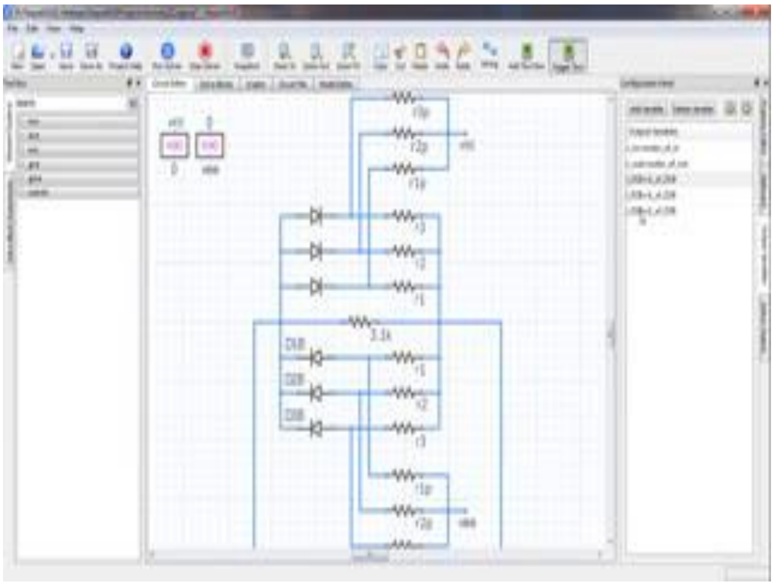
(Refer Slide Time: 21:50)



Let us now look at the quality of the sine output that we have got and we do that by looking at the Fourier components of the output voltage. So, let us bring in another output block - change the name of the file so it does not clash with the earlier one, choose Fourier equal to yes, let us look at 20 Fourier components. Output variables, let us look at both input and output voltages and one important specification is this out t start and out t end the difference between these two should be one period. So, let us choose out t end as the end of our simulation interval that is 10 milliseconds and out t start equal to 4 milliseconds less than that like that.

Let us now run the simulation and look at the input spectrum first. So, that is what we have got.

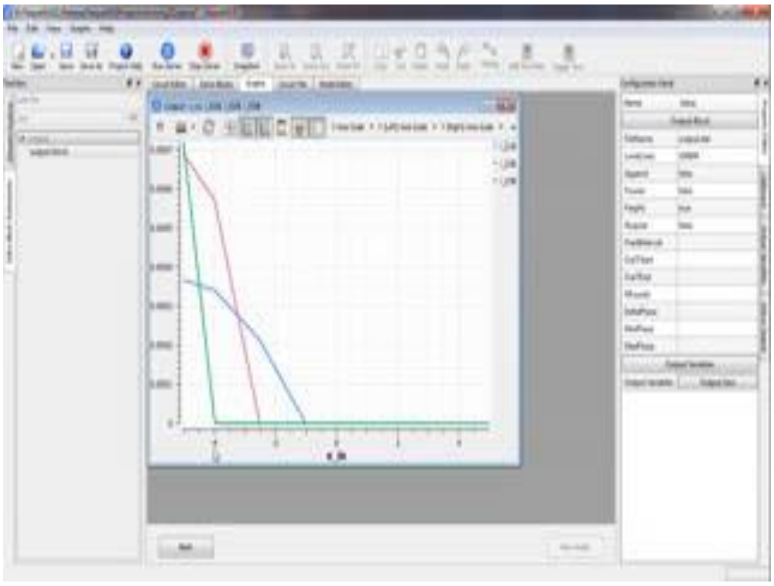
(Refer Slide Time: 23:08)



This is the fundamental that is the third harmonic, fifth harmonic, seventh harmonic and so on. Let us now look at the output spectrum and now we notice that the harmonics have almost disappeared and that is what we would really like.

Let us now look at the diode currents in particular the currents through D1B, D2B and D3B. So, what we have done is we have added three more output variables i D1B, i D2B and i D3B. Let us add these to the output file first.

(Refer Slide Time: 24:18)



Let us run the simulation and look at these variables now and what we will do is to plot these variables as a function of V in because then we will be able to see the break points like that. So, this is the current through the D1 diode and this here is the break point introduced by that particular diode and we see that it is minus 1 volt. This break point is because of D2 and that is at minus 2.5 and this one is because of D3 and that is at minus 4. So, this is indeed what we would expect and; that means, our design procedure is correct.

To summarize we design and simulated a triangle-to-sine convertor circuit and found the simulation results to be in agreement with the given specifications. You are encouraged to go one step further, hook up the circuit in the lab and look at the input and output voltages on the oscilloscope it will take some time, but it is always nice to see that what you are studying in theory works in practice. That is all for now, see you next time.