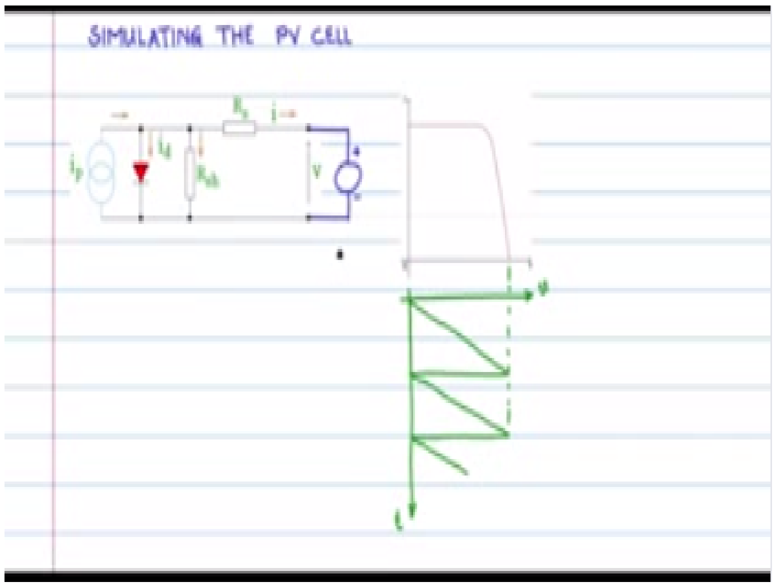
Indian Institute of Science

Design of Photovoltaic Systems

Prof. L. Umanand
Department of Electronic Systems Engineering
Indian Institute of Science, Bangalore

NPTEL Online Certification Course

(Refer Slide Time: 00:16)



Simulating the PV cell in this session let us see how we would go about simulating a PV cell before we go into the nitty-gritty of simulating the PV cell you with me in these files let us see what we want to actually do, we know that the PV cell has a model like this now this model has to be simulated and what is it that we want to see we would like to see the IV characteristic at the terminals the voltage across the terminals and the current flowing through the terminal and the external load and how is IV characteristic we know that it is something like this if we have I on the y-axis and the voltage on the x-axis the IV characteristic looks something like this.

So how can we achieve this IV characteristic by simulating this model in ng spice consider we have an external load or not like this now if we just how this external load or not like pace there will be a current flowing through this and a voltage will develop across or nota DC voltage will be there and you will have a DC current it will give just one operating point on this IV curve we need to have many operating points.

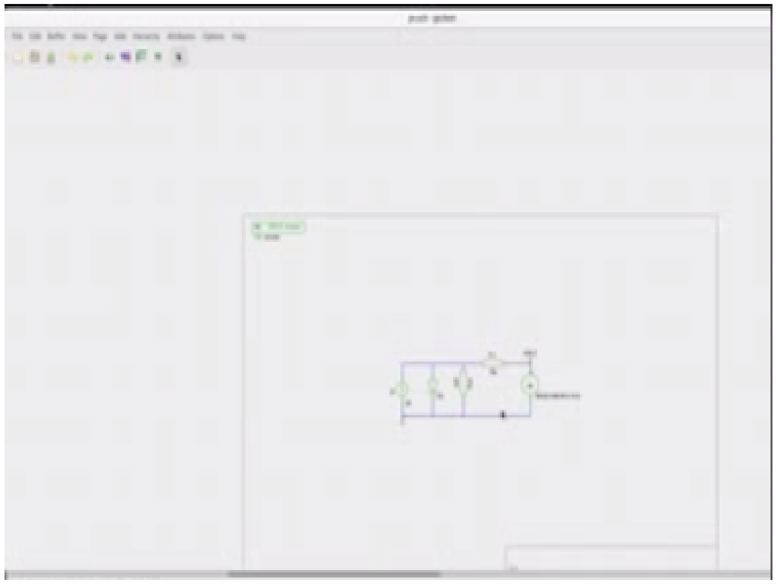
So therefore we make the load resistance R_0 variable in such a case by making R_0 0 you would achieve V_0 of zero there will be a short circuit current flowing through R_0 and when you try to increase R_0 to a larger value this I x R_0 value at that point in time would give you a voltage which would sweep the x-axis. So basically we would have an x-axis sweep voltage value and when you measure the current you would get the various points which would represent the IV curve alternately what you could also do is replace this resistance with a source incorporating a voltage source like this across the terminals will also provide us with this IV characteristic.

However the voltage source should have a time evolution in this manner let us say we have a saw tooth like this in this manner this would provide the sweat for the x-axis of the IV characteristic this voltage V and the current will accordingly vary and provided with the points for the IV characteristic what can be noted is that this wave shape for the voltage of this voltage source need not be saw tooth it could be triangle or it could also be sinusoidal as long as it is a very function of time so that this x-axis sweep is provided.

So normally we will use a sine sinusoidal voltage source here because that is easy to make an easy to implement and even in the simulation we can try putting a sinusoidal voltage source here and see the performance of this model and obtain the IV characteristic curve after we have obtained the IV characteristic curve we will try to vary RS and see what is the effect on the characteristic we will try to vary our shunt and see the effect on the characteristic and also try to vary the temperature operating temperature let say 10^0 25^0 50^0 and see what is the effect of temperature on the characteristic.

So all this let us see how we go about doing in simulation using the open source ng spice software.

(Refer Slide Time: 05:37)



Let us now go through the steps in simulating a model for PV cells in ng spice for that first let us create a new folder and it may call it as PV sim and this folder PV sim will act as our simulation project folder it is right now empty and into this all the files related to simulating the model of a PV cell will be placed. Let us go to the applications and pick up the GED a schematic editor click on that you will get the schematic editor egoism this is where the circuit schematic will be drawn maximize start and you have the MD blank schematic page.

Before you begin drawing schematic let us save this file and give it a meaningful name so go to save and then into the PV sim folder that we just created we will call it as PV . SCH and then we now have a blank PV. SCH file now into this page let us drop in the components and then connect them to form the model of a PV cell go to the add component box and here you select there are many libraries with components now let us choose the ones relevant for this particular project and choosing spice simulation elements and in that I am selecting the spice include Directive you click that here I will explain it explain about that a bit later.

Then we will choose the power rails library and that we need two important symbols one is a ground symbol without which the simulation will not proceed and then the generic power symbol it is a label we need to label the next so let us place that here I will explain that later again next we choose the basic devices library and open it up and then we need to choose few components from here one of the components we need is a current symbol for the photocurrent then we need to choose resistors the PV model has resistors in it circuit model resistors there are two symbols I would prefer the box type symbol.

So let me place two resistors there one for the PDS and another for the shunt then we need to pick voltage source, so let me pick this voltage source and place it here this is for connecting across output terminals to see the IV characteristics. Then we need to pick a diode because diode is again another component that is an important part of the PV cell model we think the diode and we have here almost all the components in place we can close this now we need to combine this into a circuit pick the current source rotate them like this pick the voltage source rotate it accordingly take the diode rotate that take the harsh in we will become our shunned or get it accordingly and then place the cds component and then next we need to wire them up before wiring you could also name the component parts.

So let us say this is ID indicating the photocurrent you would have d1 is the diode part of the model then the R shunt I can say are SH the series non ideality RS and voltage holes to be connected to the output we can call it as V_0 and the output node label we can call it as NV_0 0 node of V_0 0 NV_0 like this. Now let us go about connecting the components with the wires like this as shown after having connected the wires appropriately the circuit looks like this and here for the include directive also you can give a designator a1 and then for the file you give it right now as P V dot X will be and into this PV .SUV we will put all the models related to the circuit.

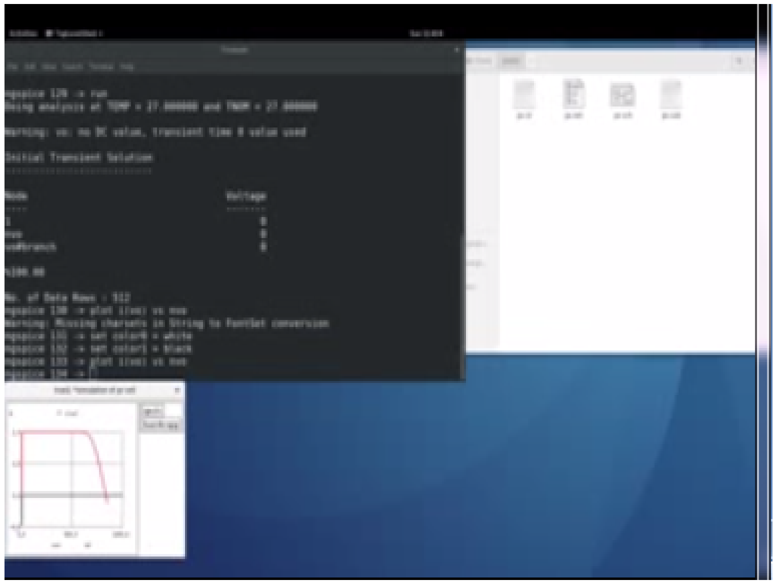
I will describe that a bit later for now just give it as P V. SUV next I have given values to the various components let me bring it to close up like this so for the photocurrent part I have given up one hand as IQ you just have to enter into the value attribute and for the resistance also into the value attribute I put one mega ohms and the series resistance point 1 ohm and for the V_0 source I have given as a sign source we need a monotonically varying source here at the output terminals so that we have x-axis sweep for IV characteristics.

I even as sign source from 0 to 0.85 peak 50 Hertz and all the parameters 0, so you can experiment with that and I am calling this node again V naught as I said earlier now for the diode I have given the value of d EF it means it is a model value and that model will be included in this PV. Sum, so this now completes the entire schematic. Now we see that in the pv sim folder the schematic file P V. SH has been created you can always double click on that and then see the circuit and edit it at any time that you like.

Now you should note that ng spice does not directly use the dot SCH file for simulation it has to go through one more step of conversion to a net list and ng spice can only recognize and use net list for simulation. So now what we will do is to create a net list before creating the net list there are two files that needs to exist in this folder one is the PV. SUV rim recall that we had mentioned PV. SUV here in the spiced include directive we need to provide that file even if it is a blank file and the other one is P V. CLA or the circuit file in which will contain all the analysis that ng spice is supposed to do.

So to create these two files let us go to a text editor and just a blank file we will save it into PV. PV same as P V. s Q P now this will save and you will get one blank text file a V.SUV we will also create one more file and name that as PV. CIR which will contain all the analysis that ng spice is supposed to do. Now you have the three files PV. CIR, PV. SCH and PV. SUB. Now using the SCH and SUB file we can now compile and create a dot net PV dot net.

(Refer Slide Time: 15:33)



Now let us see how we go about generating this net list file for this let us go to terminal open a terminal window go to the simulation subfolder and check that we are the right fold of the right files now here let us type in the command for generating the net list G net list – G spice - SDB output- pv. Net input dot SAH, so this is the command for generating the net list and it would have generated the net list here so see that here new file has been created that fold at the fourth file which is PV dot net.

So if you read through the P V dot net list file open it the initial statements are automatically put in by the G net list program see here the first file include TV. sub this is coming from the spice include directive and this or the net list of the circuit that we put in into the circuit pv module circuit diagram I am going to draw your focus to this point D 1 the value that we gave was the EF it is the name of a model right now it does not have a definition because PV dot sub is blank so we need to define this DES model within P V. sub then the net list becomes complete.

So let us do that and then recompile and generate the net list have close this and open VV dot sub so that is open the first line is normally a comment line but you put in some comments so let me type in something like the D of whole name something meaningful then it is better that you have some demarcation star would indicate comments. So let us say class of diodes against are all these are comment lines now starts the model dot model the name of the model which we gave BEF a diode model within these parentheses and then just close it indicating now all others are comment line all the line starting with the star our comment lines the dot model D within pare in thesis is the model default diode model that you put.

If we want to have any specific special diodes we may have to fill in the parameters within this parenthesis that you can look into the data sheet manual or spice and then appropriately edit the models. Now for the for now we will put this default model and save that is saved we close this and then we do the generation of the net list, so when we redo the generation of the net list the new necklace would have been created in PV. net it would have been updated just see that with respect to the previous one it has been updated there is a Model D which has been now included it is residing in a V dot sub.

So D the diode DF is now defined here and then all the portions of the net list are fine and okay and it is supposed to execute well in the simulation. So now we have a full-fledged proper net list we need to put in the analysis within the dot ca5 so let us see what we will write in this dot C AR file now open PV dots here it is a blank file first line is again a comment line I will just type in simulation of PV cell I relieve one more line and then put in the analysis I will put a dot Tran transient analysis with print step of 0.01 milliseconds 5 milliseconds as the end of time and you I see use initial conditions.

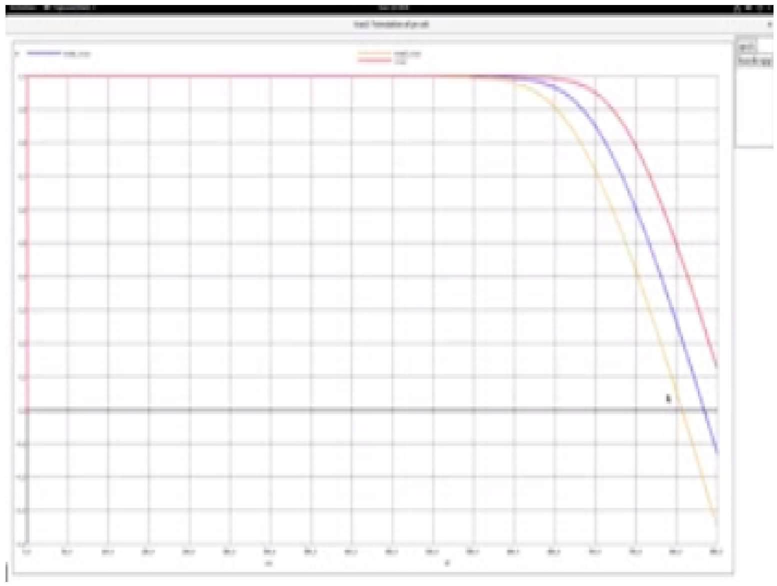
I will also include dot include T V. net that is we include the P V. net into this okay this 5 milliseconds is coming from the schematic where if we go to the schematic you see that we have an output which is a sinusoidal source 0to 0.85 peak this is 50 Hertz which means 20 milliseconds period so in a quarter of this period it would have reached 0 to 0.85 in 90^0 time.

So it would have reached the full sweet value of the x axis so that is why I have given 5 milliseconds here that should be sufficient save this and that is it we have all the files here ng spice will use this and these two files it we will be using this will in term called PV net which will in turn call VV. Sub. Now let us simulate the circuit file you go into the terminal window type in ng spice P V. say R that is it for you are now into the ng spice environment you type a one run will execute the simulation you see that now there are a financial data course and these are the vector available to you node one node nvo and V_0 branch current.

So now you recall why we give the label so that you can recognize which node that particular node label will appear here for the vector name and we can easily recognize that particular node to the circuit and then use it for plots if you plot the branch current IV_0 vs. can we go, so it will give you the plot of the IV plot of the PV cell. So this is the V_0 the voltage across the terminal and this is the current through the system you know this is a short circuit point and this open circuit point.

So you can expand it and then see that this is the short circuit point one amp we had given it as one amp and then this point at I equal to zero this will be the open circuit voltage point and somewhere here would be the peak power point okay. So this is how you would get your plot outs now if you want to get a plot out on white background this is what you need to do set color zero representing the background as white and also set color of the foreground one as black and then plot again plot IV_0 current will be V_0 versus the maximize it and then you see that it is so you could choose whichever type of background and foreground that you want according to your convenience.

(Refer Slide Time: 24:24)



Now opening the schematic file and zooming in the circuit and let us say we would like to see this at all IV characteristic curves with RS as a parameter as RS varies so let us see how we go about simulating with IV characteristic of the different values of RS. So now open the terminal window going to ng spice VV dot c area so now you are in the ng spice environment you can use listing and see what is the net list as we are that have been loaded, now what we want to do is we want to change the parameter of the RS per series resistance.

So we would say a point one point three point six owns just for example right now so how do we go about doing it is we will execute this run of the simulation with RS is equal to zero point one so you just type in to run so one set of simulation is executed these are the node voltage the branch current that you see if we want to see the vector names just put in display. So when you type in display you see these are the vectors available to you the voltage of node one the voltage of the label denote the time and the current through the branch V_0 this all these vectors are grouped into one called tran1 group when you type in set plot you will see that tran1 which is the current plot group okay is named tran1 it is a simulation of the transient analysis.

So now if you alter the RS value to zero point three ohms let us say now the RS value is altered and now you run the simulation now I get another set of vectors here I have been set plot just to show to you that now you have a tram two simulation with the second simulation so I still have the vectors of tram one I now have the vectors of Tran two now let us say that I also alter once again RS to 0.6 ohms and run the plot and I will type in set plot you will see I have Tran 3 all the vectors corresponding to transit.

So each of these plot groups will have the vectors of all the node voltages and currents now let me shift the current plot to Tran one that the hotspot at R is equal to point one you can check that again type set plot you will see that the current plot now is Tran one now I will shot before plotting let me set the background color white set the foreground color to black and now plot now what are the things that are plot I have to plot the IV characteristics the x-axis I will take it always corresponding to the transient analysis one and the y-axis is the one which would vary for the other for all the other transient analysis at RS is equal to 0.3 and 0.6.

So plot current V_0 versus NV_0 the voltage across the output terminal labeled then turn to current to the terminal vs. NV_0 naught same speed then Tran three current versus the same sweep, so when you plot you get the three sets of plots like this maximize it and you will see now you see this red one is I V_0 this is tran1at RS is equal to 0.1 ohms.

Now blue one Tran 2 is for the case when R is equal to 0.3 this is for the case when RS is equal to 0.6 and it is also agreeable with the theory that we had discussed earlier and the other non ideality causes quite a lot of distortion as it increases so quit this and you can exit out of the ng spice environment. Sometimes you may want to run the analysis repeatedly many times and then every time for you to type within the ng specie environment may be cumbersome so what you could do is now let me go into the ng spice environment.

Now let me type history you know the history of commands that you have done previously you can now go to PV. CIR open that and let us put in the control scripts so between control dot control and dot n see you can put in all the scripts that you had typed here and whatever is relevant.

So let us say we wanted to first run the simulation with RS is equal to 0.1 then we would like to alter RS to 0.3 and then run the script then again alter RS to 0.6 volts and then run the script now we have three analysis two and one and two and two and three we would like to set plot to run one then we did a set color of the background to white set color of the foreground to black and then we plot the graphs so what is the plot so I what is the here I could probably copy that and then paste it here.

So this is the analysis that we wanted to do and plot, so this I will save it and then close it so now when you call ng spice it will execute all these control statements and then provide you with the final results. So let me exit out of it ctrl L or clean-slate call ng spice PV. CIR. So it will go through the complete set and execute and give you all the three results so this will help you to do repeated simulations with minor changes so you could make the changes directly here in the control statements this will save on iteration.

In a similar manner you can perform the experiment for any other parameter in the circuit you could probably do it for our shunt although the variation of the IV characteristic curve for or shunt variation is not as large as significant as the series no ideality RS I would like to show you the variation of the IV characteristic with temperature it is I was like a different in the sense that the command is slightly different.

So let us have a look at that we opened PV dot C area now we run the simulation of the circuit as it is normally default temperature is at 27^0 centigrade so this simulation run tran1 would be a 27^0C we change the temperature here but changing the temperature for the cycle is using the option statement M equals let us say 0 t 0^0 it is at0^0 and from the templates run that and then again we will change the temperature to50^0 and then around that simulation then we set the plot to this is some logic that you need to use you see that when the when the temperature when the temperature is lower we will be higher and when the temperature is higher we will see is lower.

So we could probably take the one with zero as the reference so which is transient 2 so let us set the current plot to transient 2 and then we are setting the foreground and background black and white and I_0 versus NV_0 is transient 2 because we are set it as the reference you can then set transient analysis 1 I_0 versus NV_0 with respect to two tangent analysis 3 with respect to 2. So this would execute and give you the temperature variation in the IV characteristic.

So let us call this ng spice V V dot C R so that will go and execute expand the simulation then you see that you have the variation of V_0 with respect to now here if you see the red line I_0 is the one with respect to 0^0 that is transient analysis 2 transient analysis one at 27^0 so it is slightly reduced and transient analysis 3 this is at 50^0 so still further reduce.

So as the temperature is increasing you see that the curves start going with in V_0 starts decreasing and that is as per our understanding 2 as per the theory also, so like that you can make any analysis using ng spice this is a sample that I have said there is lot of scope for expanding and checking out various things and various parameters I will leave all that to you.