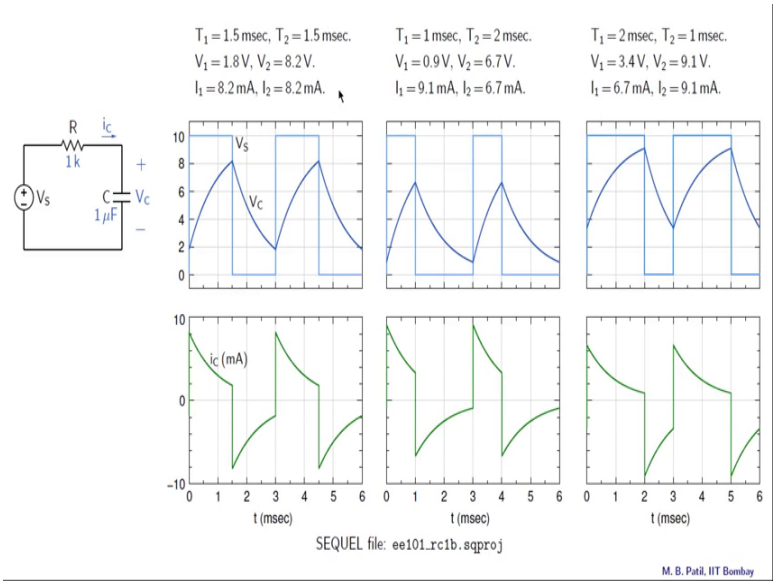
Basic Electronics
Prof. Mahesh Patil
Department of Electrical Engineering
Indian Institute of Technology, Bombay

Lecture – 12
Stimulation of RC circuit

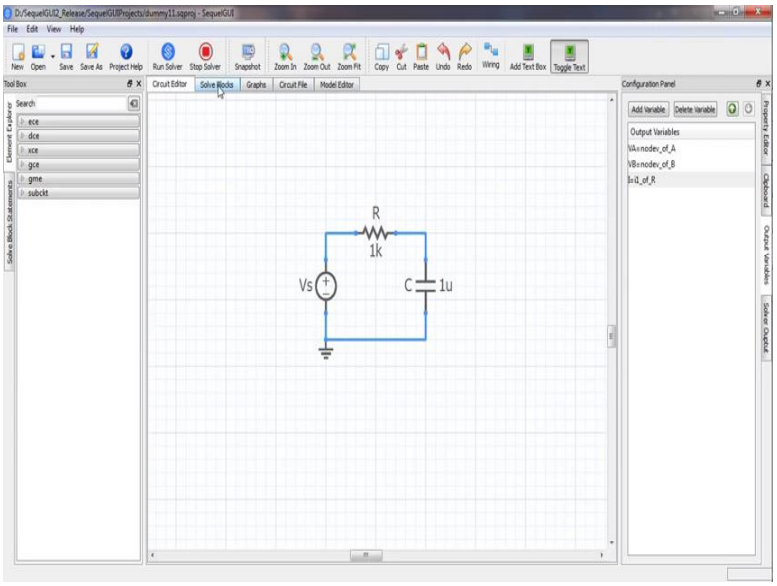
Welcome back to Basic Electronics. We have looked at a few RC and RL circuit examples we will now simulate one of these circuits and check whether the simulation results are in agreement with our theoretical expectations let us start.

(Refer Slide Time: 00:34)



In this circuit simulation session we will simulate this RC circuit which we have seen earlier, it is a simple series RC circuit with R equal to 1 k, c equal to 1 micro and V s is a square wave voltage, wave from 0 volts to 10 volts this interval in which V s is high it is called T 1 and this interval in which V s is low it is called T 2. Now there is the readymade circuit file for this example, but we will not use that what we will do is we will start from the beginning, see how this circuit schematic can be constructed, how we can specify the simulation parameters such as the time step, and how we can specify the output file and finally, how we can observe these plots and once we set up this simulation project we can change the values of T 1, T 2, RC etcetera and then look at the results.

(Refer Slide Time: 01:38)



We start with the sequel GUI to release directly, and under this there are several directories and files and if we go down there is an executable file or sequel GUI dot e x e. So, we double click on that and the GUI starts and then we have these 3 windows left window called the toolbox, then a central window which does various things and we will see what it does and then this is the right window called the configuration panel. The first step is to get the components and let us start with the register, on this left side in the tool box we have the element library and here we see something called e c e that stands for electrical compound elements and that is where our register is going to be. So, we click there then we click on linear because the register is in this group and then we find the register over there. So, we single click here and then we double click here to get the register.

Now, let us get the capacitor and use another method to do that, suppose we know the name of the capacitor element and that is c dot e c e, when we type that in the search box all elements which have that string in their names will show up here and that is our capacitor. So, we click there double click here and the capacitor appears over here. Let us zoom in a little bit and we can do that by clicking on this box here like that or we can also use the wheel of the mouse and zoom in or out like that. Now we want to rotate this capacitor, so we select it with a single click and then type R then it turns.

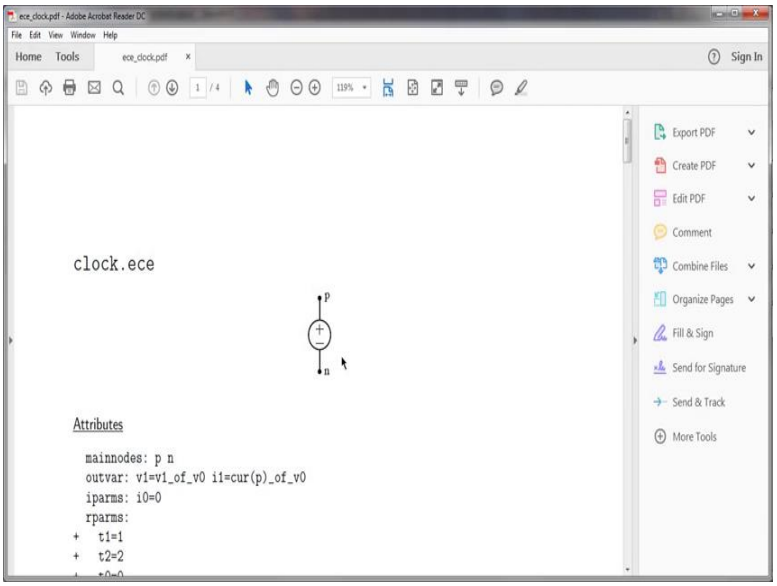
Let us now get the square wave voltage source and that is called clock dot e c e, and in addition we also need a reference node because the simulator uses the nodal analysis method internally, and it needs to know which node is the ground or the reference node. So, now, we need to get the ground element, which is called ground dot e c e and there it is. Now they have all the required elements and we can proceed with the wiring.

Before we start our wiring let us arrange the components a bit more neatly, how do we move components for example, this capacitor we can do 2 things; one click on it keep the left key pressed and then move the mouse like that, or we can select it with the left key realize the key and now use the arrows like that. So, that is where we want the capacitor to be and let us similarly move this ground as well like that. Now let us start over wiring to start the wiring mode we can press w, and this wiring mode gets highlighted and now we click on any node like that and now we are in the wiring mode; we can take the cursor where we want click here to introduce a bend in the wire, and now click on the destination node and when we do that the wire is completed.

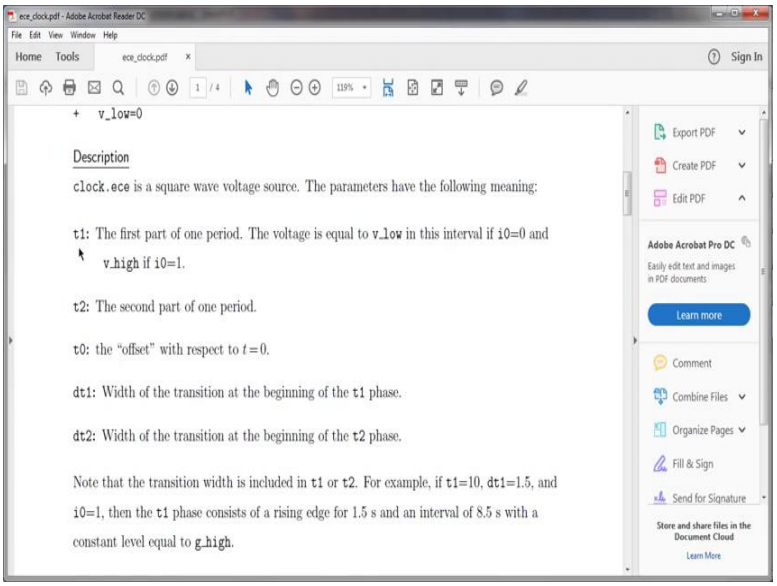
Let us complete the rest of the wiring then press w, complete this wire instead of pressing w we can also click here to enter the wiring mode, it does the same thing. Now the wiring is completed and now we want to change the component values for example, this resistance should be 1 k; this capacitance should be 1 micro ohm. So, we click on the resistor a single left click and here are the real parameters, and the parameter of interest to us is this r the resistance value, right now it is 1 we can make it 1 k, and note that there is no space between 1 and k we can also write 1 e 3 or 1000 they all mean the same thing.

Next let us set the capacitance value to 1 micro ohm. So, we click on that and here is the capacitance we change that to 1 q u stands for micro ohm we can also write 1 e minus 6. Now let us come to this clock the square wave voltage source, and we find that it has got several parameters T 1, T 2, T 0 etcetera and to look up the help document for this clock, we right click on it and choose help over here.

(Refer Slide Time: 07:15)



(Refer Slide Time: 07:21)



And then we see this p d f file and that describes the meaning of the various parameters for example, t 1 the first part of 1 period, t 2 the second part of 1 period and so on.

In our case t 1 and t 2 are going to be equal and each one is going to be 1.5 milliseconds; milliseconds is written with a small n, t 0 is not relevant, b low should be 0 because our clock is going to go from 0 to 10; v high should be 10 volt now d t 1 is the rise time or the fall time depending on the value of this i 0 and we will make both of these equals and we will make it pretty small compared to t 1 and t 2 . So, we will make it 0.01

millisecond like that. We can also change the names of these components and that is really optional because the program does generate some names by default for example, this register is called e c e underscore R dollar 0, but say we want to change that to R then we click on it change it to R here, similarly we can change the capacitor name to c and this source name to V s.

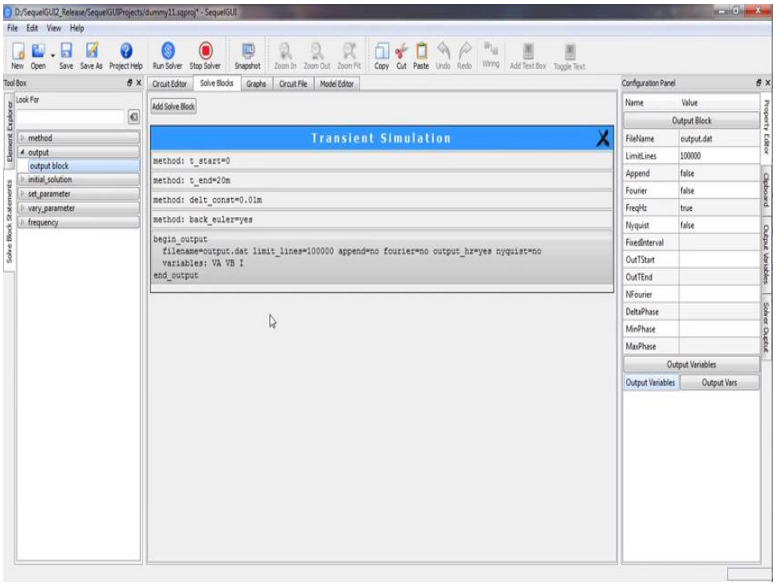
What about nodes? This node for example, is e dollar 5 right now, suppose we want to change it to say A we can do it like that, what we can do is we can also change the node as a property of this resistor for example, if I click on this register the node names come over here and we already typed A over here, that is the P node of the register that is why it is showing up as P. Now the n is still some default n and we can change that let us say to b and now we can actually check the names of the nodes we just take the cursor place it on top of the wire without clicking and then the node name shows up; this one is A and this 1 is B.

Since this is the reference node we can give that the name 0 although it is not ready required. Now suppose we want to see the values of these components on the screen, the way to do that is right click on the component and say add default property text boxes, then the property the default properties that is the resistance in this case shows up here. We can also show the caption of this element that is the name of that element by right clicking again and selecting add caption over here. So, that is R we can move this around. So, it looks a little neater like that.

Our next step is to indicate to the circuit simulator, which output variables we would like to keep track off for example, for plotting purposes as the simulation runs. So, what are the variables that we are interested in? One is this voltage here that is the source voltage, or the node voltage at node A ith respect to ground another quantity of interest is this output voltage or the voltage at node B with respect to ground and the third variable of matrices t is the current either through the register or the capacitor. Now to select these output variables, what we need to do is to click on this output variables tab and then on add variable, and when we do that the cursor becomes a plus sign and now we are ready to select output variables; for example, if we click on this wire notice what happens what happens, now is this node V of a which means node voltage of A appears here as an output variable.

Similarly, let us select the node voltage at this node which is node B, as another output variable like that and also the current through this register as the third output variable. So, we click on the register and select I 1. So, I 1 is the current through the resistor; now notice that the simulator has generated some default names for these variables var 2, var 3, var 4 now these names do not really make sense to us. So, what we will do is to change these names to some meaningful names for example, this name can be V A this name can be V B and this name can be just I.

(Refer Slide Time: 12:47)



Our next step is to prepare something called the solve block. Now the solve block test the simulator what type of simulation it should do, and what are the simulation parameters what output files it should generate and so on. So, let us add a solve block by clicking here and by default the solve type is DC simulation. In this case we do not want this, we want transient simulation. So, we click on this bar single click and then we go to property editor, click here and then choose transient simulation from there.

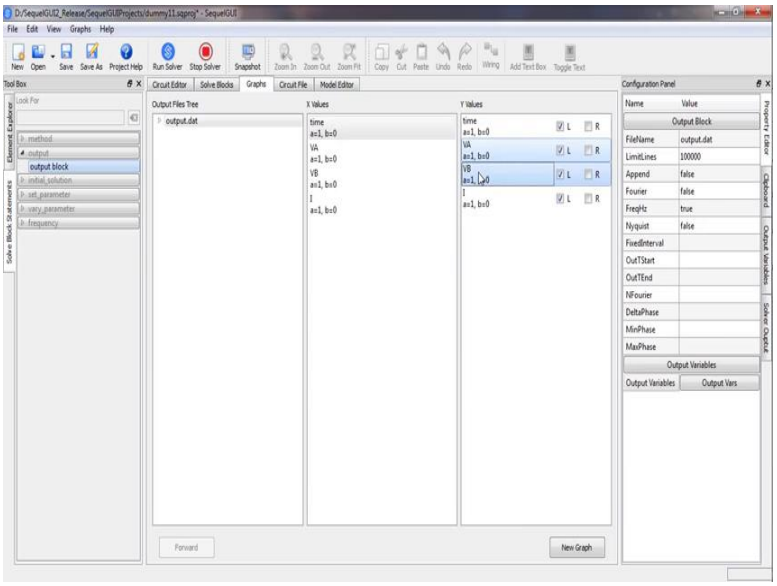
Now, as soon as we do that several things come up here, t start is 0 by default and we need to fill in all of these others what about the end point of the simulation t n that we need to decide upon. Now our time constant is R times c which is 1 k times 1 micro ohm that is 1 millisecond and we should choose a time here, which is reasonably large compared to that time constant. So, let us choose 20 milliseconds. So, that is 20 small m without any space; what about the time step that is specified by this delta t constant here.

Now, the time step the simulation time step it should be small compared to the time constant; our time constant is 1 millisecond so we can specify a reasonably small time step like 0.01 millisecond. There is also this method called here, which says backward Euler equal to s, there are several options for the transient simulation method and in this case backward Euler works fine so we will leave it at that. Now one last thing and that is the output file, we need to tell the simulator what output file it should create and which output variables it should store in that file.

Now, that is done by selecting this output block from this output tab. So, we select that keep the left button of the mouse pressed and drag and drop it over here, and notice that this statement the output block statement has expanded into a few things here, and if you click on that, we will be able to edit those on the right hand side here in the configuration panel. Now in this particular example the file name is output dot that and it can stay as it is, and the only thing we need to really change in this particular simulation is these output variables, that it is this field over here and when we click on that double click on that we get all the output variables which are available for storing, and you notice that these are the same output variables that we created V A V B and I.

So, let us choose all of these and say and when we do that notice that those variables appear over here, and with that our simulation setup is now ready and we can run the simulation, that is very easy to do all we need to do is to click on this run solver button here.

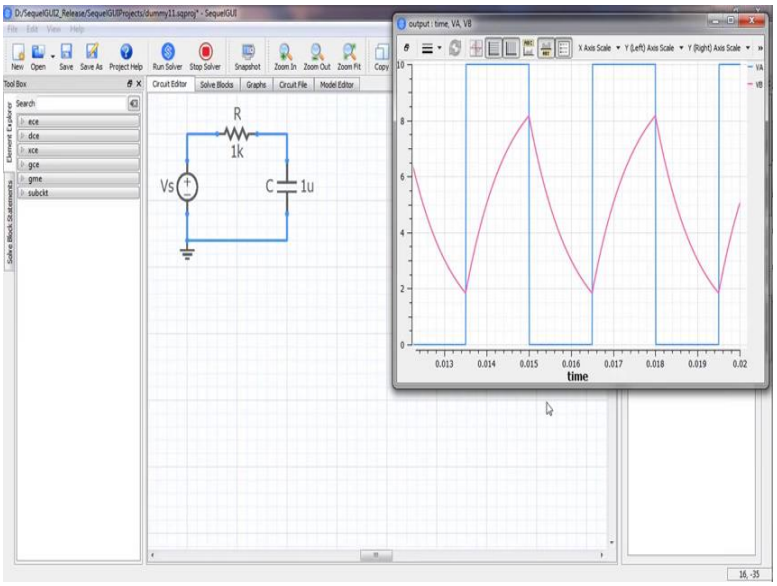
(Refer Slide Time: 16:39)



So, the simulation is successful and the control has automatically been transferred to this graphs menu. Now in the graphs menu we see this name of the output file, which we have seen in our solve block and when we click on that, the variables which are stored in this file appear over here this stands for the x axis this column, and this other column stands for the y axis. Now notice that we selected on the V A, V B and I in the output file, but because it is transient simulation time gets automatically selected by the program and now what we can do is select the x axis variable and y axis variables and then generate a plot.

So, let us select time as the x axis makes sense, and what we will do is select both V A and V B as the y axis. Now V A is the input voltage, and V B is the output voltage or the capacitor voltage.

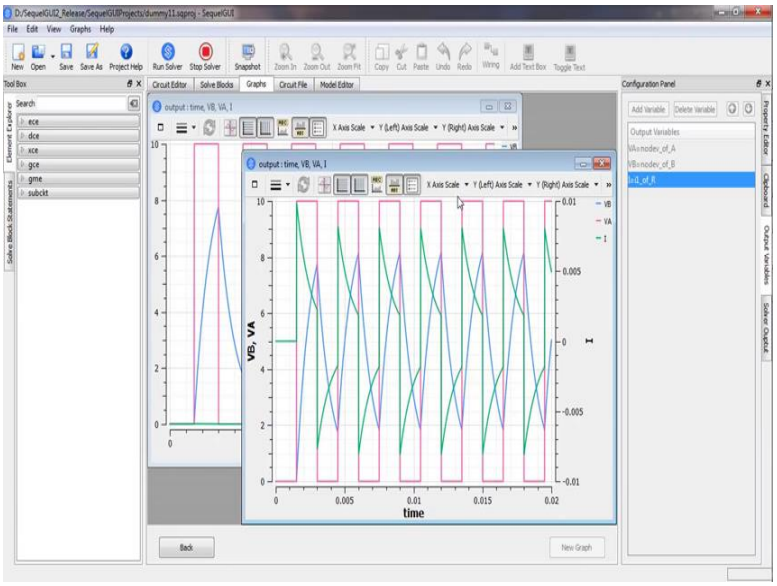
(Refer Slide Time: 17:59)



So, let us see what this look like after selecting we click on this new graph button, and that is what we get and if you do not like this line thickness you can increase it like that. So, this blue one is over V A marked here and the red one is our V B and we can expand this because that gives us our steady state and we can look at it more closely like that. We can look at these values the high value here and the low value here and make sure that they match with our expected results, which we discussed earlier and we can also get the value by clicking at a given point like that then that tells you the x value as well as the variant value. So, the var value here is 8.2 volts, and the var value here is 1.8 something volts.

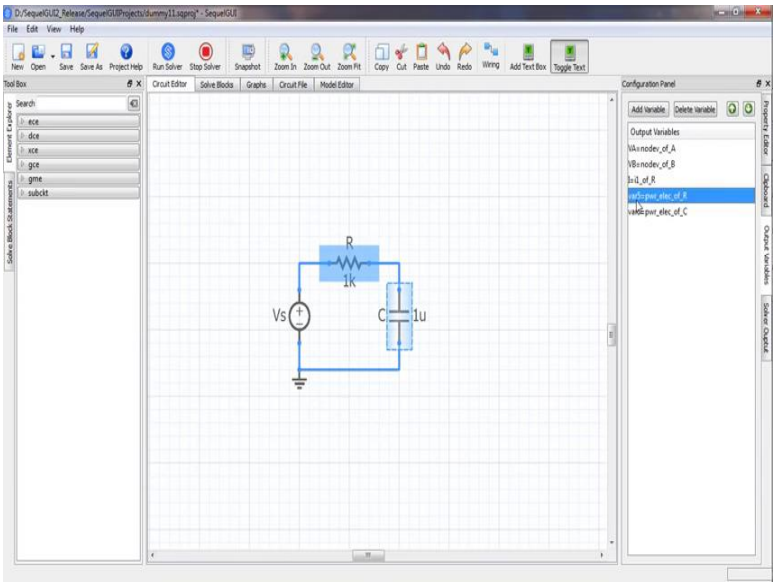
Now, suppose we want to see this graph along with our circuit what we can do is click on this button called dislodge and then the graph window becomes free, now we can go back to circuit take the circuit over there maybe, and bring the graph also in the same flame all right and now we can see this together.

(Refer Slide Time: 19:43)



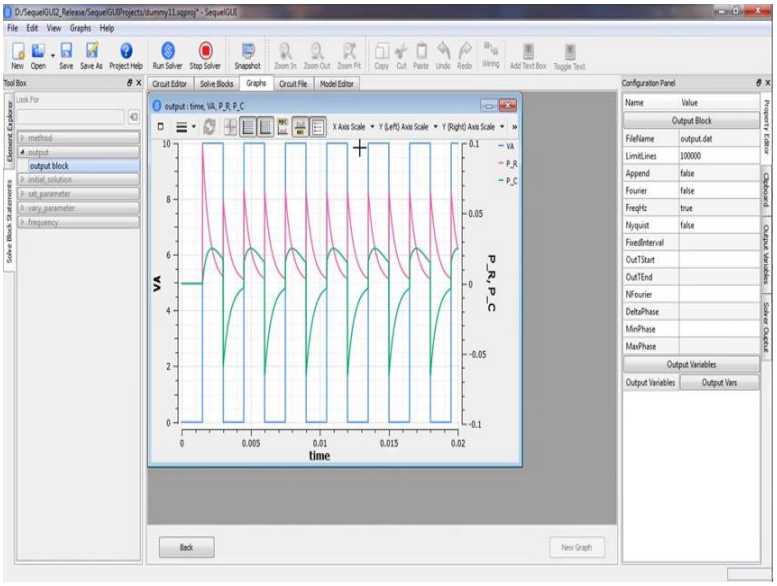
Let us now look at V A V B and the current together. So, we selected the current and now we click on new graph and there is a problem because we really cannot see the current here, it is too small because it is going to be in the milliamps range. So, therefore, what we can do is to go back and select the right y axis for the current, and then plot again and now you can see that we can resolve the voltage as well as the current. So, let us expand this steady state part. So, this one is our input voltage that is our output voltage and the green one is the current V A and V B are plotted with respect to the left axis and I on the right axis. So, this value for the current for example, can be rid off from this axis here.

(Refer Slide Time: 20:48)



Now, what we will do is look at the power absorbed by the register and the power absorbed by the capacitor. So, let us add 2 more variables click on the register and choose power electrical here. Similarly click on add variable click on the capacitor select power electrical, let us give some names to these powers let us say we call this P R and this as P C we now need to go to the solve blocks tab and add these new variables new output variables to the output file. So, we click on the output block and go to property editor here, go to output variables and now you see the new variables there P R and P C. So, we select those and say.

(Refer Slide Time: 21:56)



And now let us run the program and let us see what these powers look like. We will also plot the input voltage for reference and since the powers are going to be on a different scale, let us choose the right y axis for the powers. So, that is what it looks like, let us look at the steady state part more closely. So, that is what it is; here the input voltage is higher here the input voltage is 0, so the capacitor is charging in this interval and in this interval it is discharging.

Now, this line is the 0 of our power, above that we have positive power below that it is negative. Now notice that when the capacitor is charging the power absorbed by the capacitor the green one here; where p is positive and that is to be expected because the voltage source is supplying power to the capacitor and also to the register of course, in this interval the capacitor is discharging; that means, it is delivering power to the register and therefore, the power absorbed is showing up as negative and notice also that these 2 powers the resistor power and the capacitor power are equal and opposite and that is of course, because the source is 0 during this interval. So, the capacitor delivers power and the resistor absorbs all that power.

To summarize we simulated a simple RC circuit from scratch, we have learned how to pick the various components make connections set parameter values and plot the various quantities of interest as a function of time. You can follow this procedure and simulate other RC or RL circuits and check whether your analytic results agree with the simulation result that is all for now, see you in the next class.