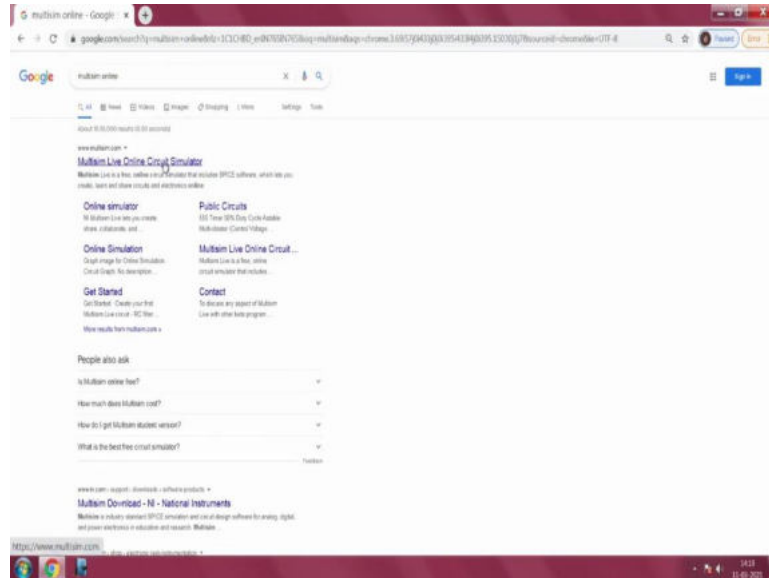


Mathematical Aspect of Biomedical Electronic System Design
V S N Sitaram Gupta V
Lecture 31
Multisim Simulations for Biomedical Signal Conditioning Circuit

(Refer Slide Time: 00:33)



Hello all, welcome to the module as a part of TA sessions I will be discussing on electronic circuits that is required for designing a signal conditioning circuit for the sensor. So, this complete TA sessions will be divided into 3 phases. In the first phase we will be discussing how do we use an simulation software, electronic circuit simulation software in order to build the electronics and analyse the electronics using the software.

In the second video I will be discussing on how do we design a signal conditioning circuit for the sensor as in the previous sessions it is it has been already discussed about the sensor and different sensors have been taught and for that particular sensor how to design a signal conditioning we will see.

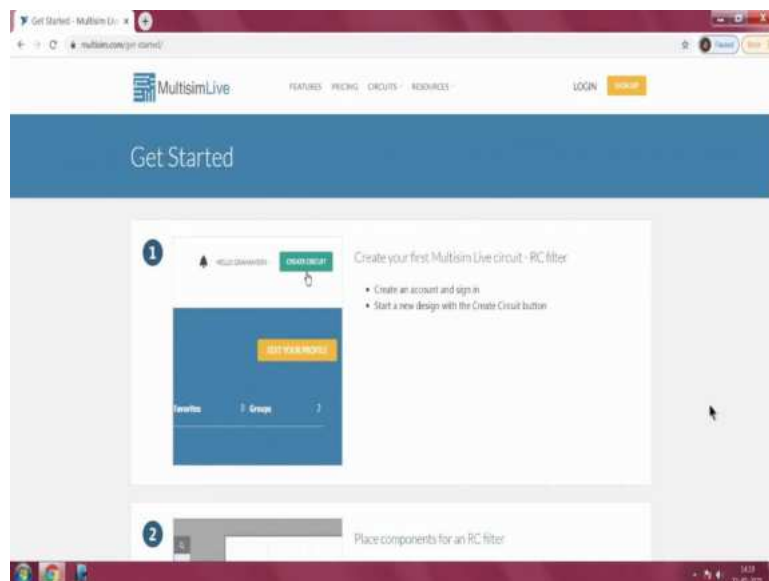
Whereas in the third video, we will discuss on how do we interface the sensor to the data acquisition device such that change in the any variations happen in the sensor will be acquired and recorded processed then will be displayed to the user as user may not understand or the end user may not understand what complete process happening inside the system. And it is always important to display the data were user has to understand about the input parameter we have to do some kind of processing.

So, in order to do that we will be using a data acquisition where we will be interfacing the sensor and the related electronics which is nothing but a signal conditioning circuit to the data

acquisition and we will read the data and we will be displaying it on a GOI based app. So do not worry all these videos will be maximum of 30 to 40 minutes each video will be of 30 to 40 minutes length.

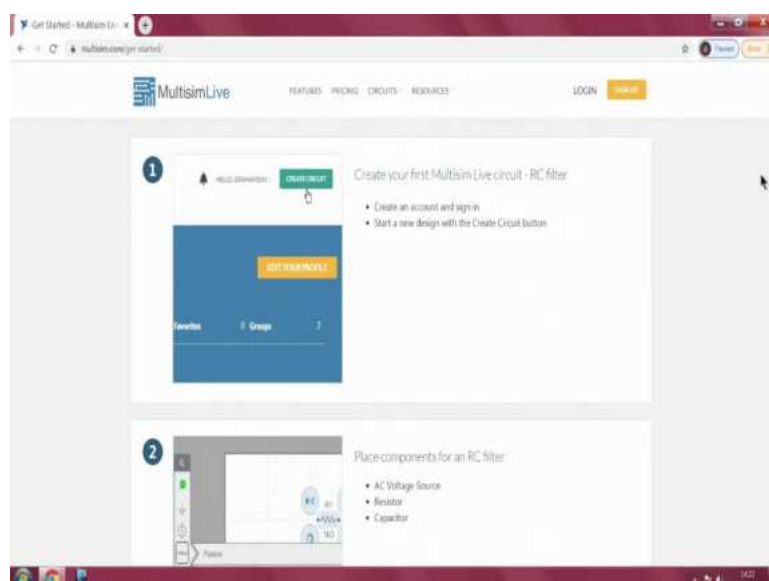
So, I will be starting with how do we use a circuit simulation software in this video. Of course, there are different circuit simulation software's available, but I am going to use online software which is multisim. So, in order to use this software just go to multisim.com.

(Refer Slide Time: 02:46)



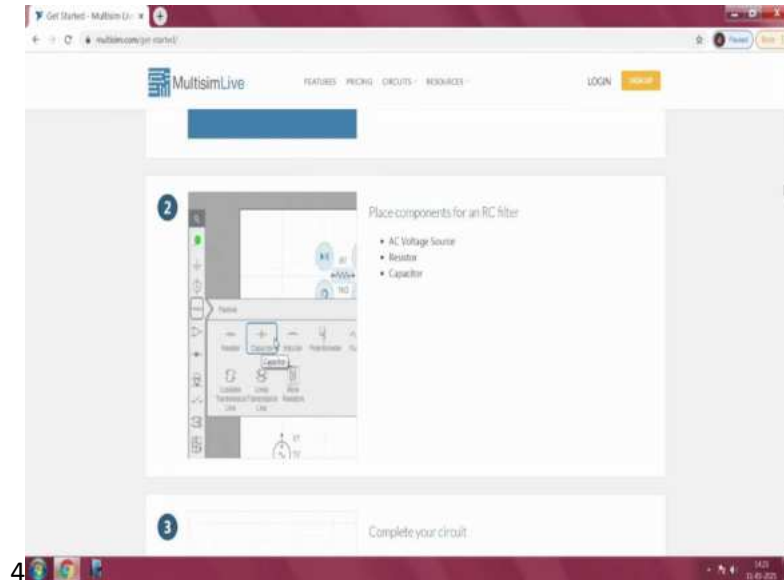
So, in order to understand make you aware of how this circuit simulation software works, you just go to the resources there we have option called get started. So, in order to analyse any circuit, it has a total of 6 steps.

(Refer Slide Time: 03:02)



First step, just create a circuit where you have to sign in where we should create an account in case if you do not have. Then just sign in into the account then just click on the create button.

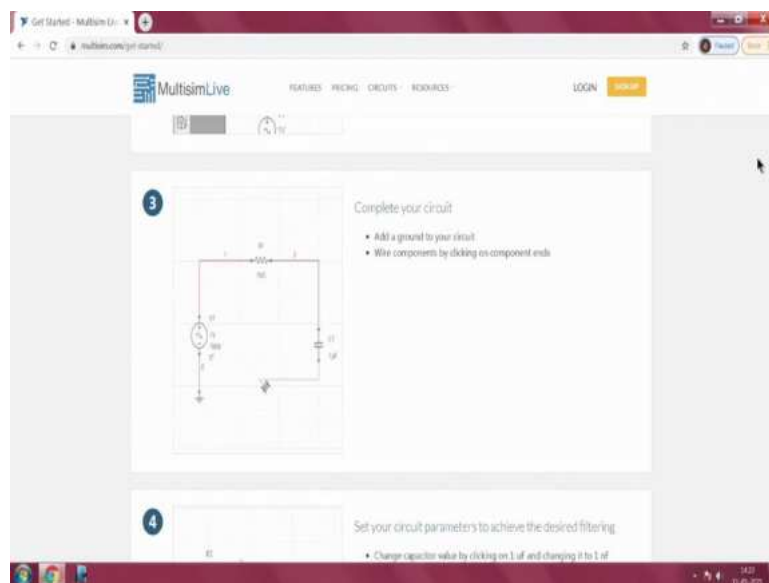
(Refer Slide Time: 03:15)



The next, next step is that, whatever circuitry of planning to realise place it on the component area just by simply dragging and dropping there. So, we have there, they have a database a basic database available on to the online multisim software such as say if you want to realise a simple RC circuit or a simple resistor divider network circuit.

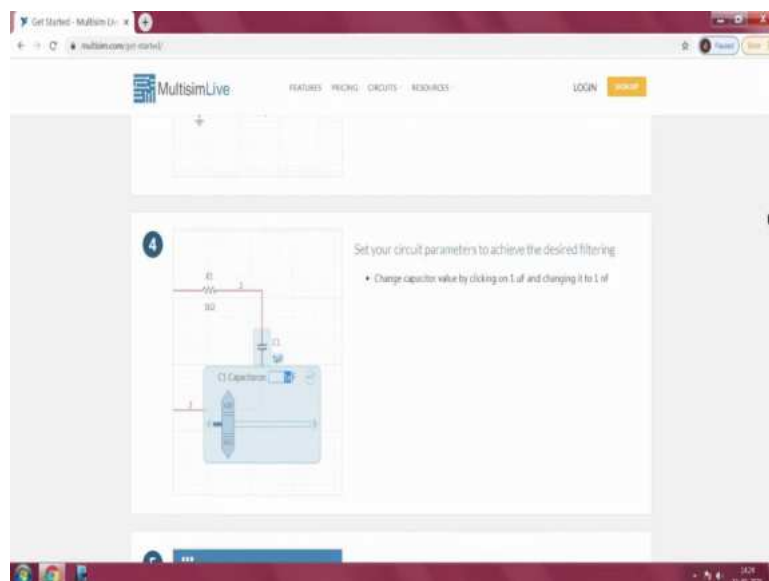
So, you have to place a component of have to place the passive components like resistors capacitors, everything along with that you also have to add a voltage source and the ground and all these components will be available in the database. Just select the proper component which is required to realise your circuit and drag and drop it on your window.

(Refer Slide Time: 04:09)



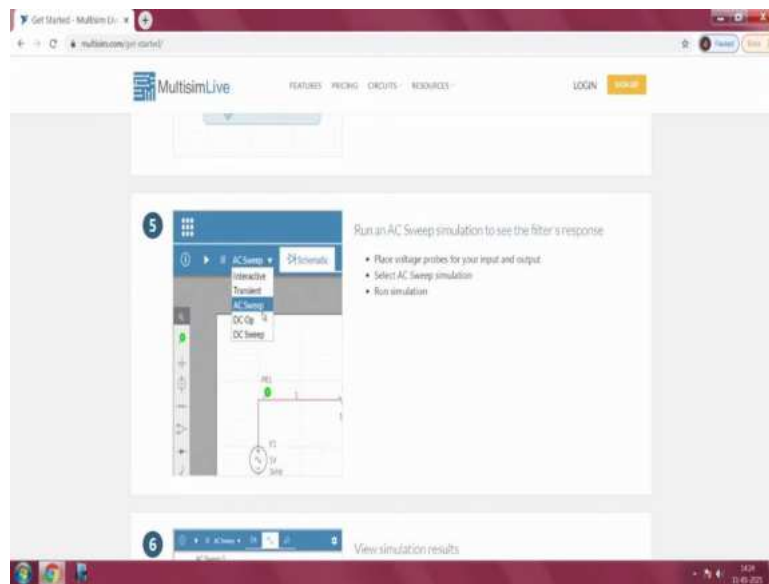
Then to just complete your circuit by wiring the all the components. So, by using a wiring tool.

(Refer Slide Time: 04:19)



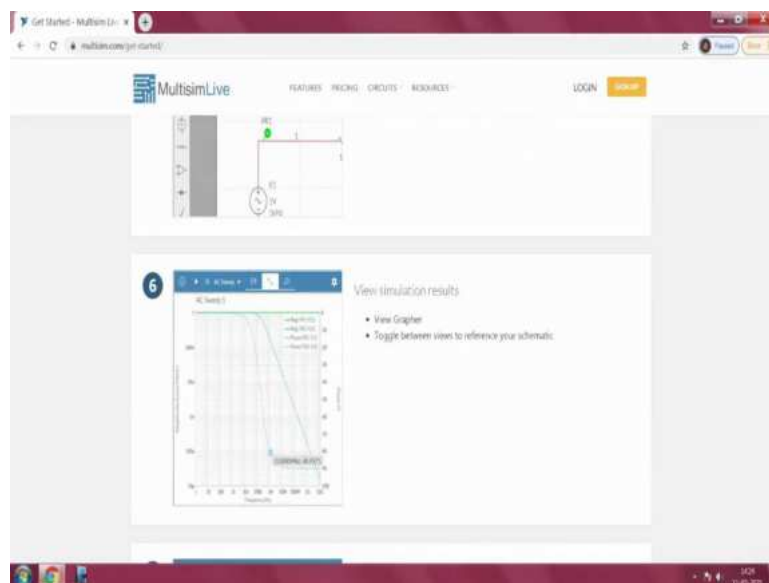
Then after wiring, either you can select the values of the passive components while you are placing it or after wiring you can select the values required for all the passive components or as well as the devices that you have placed on the window.

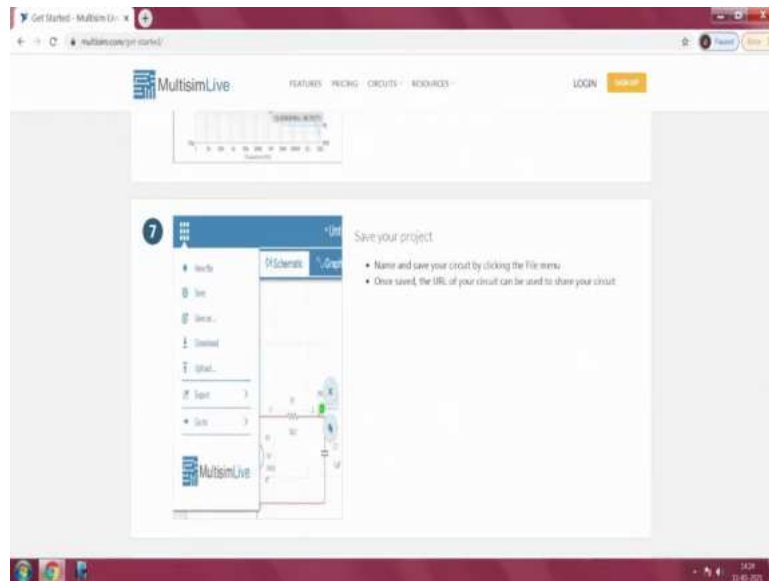
(Refer Slide Time: 04:44)



Just by clicking it. After the selection of values, select proper response, whether it is an interactive response, just one time response, transient response, AC sweep analysis or DC sweep analysis.

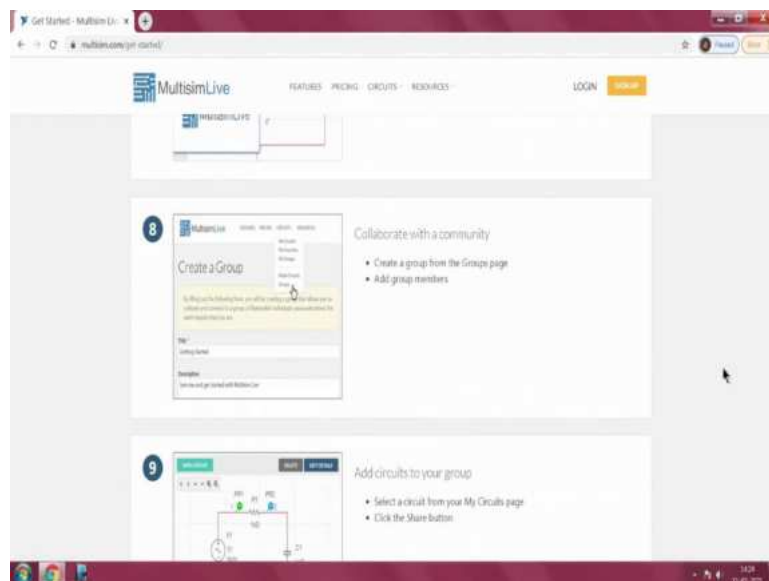
(Refer Slide Time: 05:05)





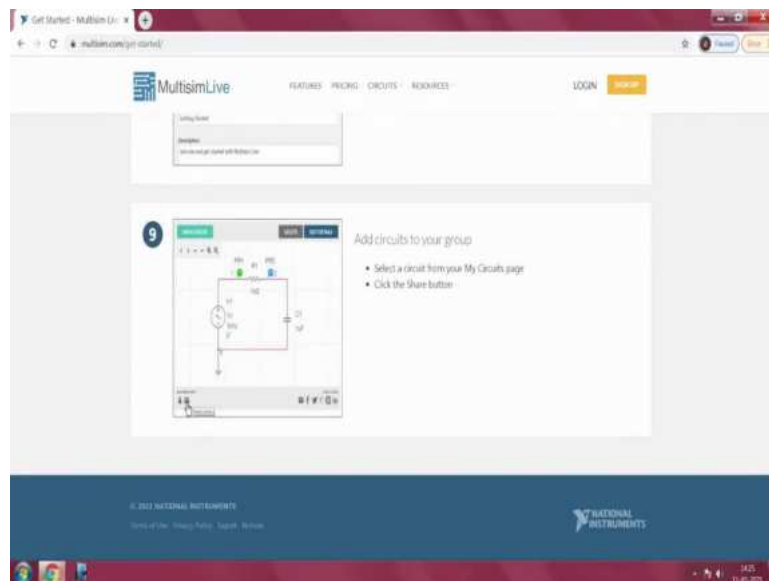
Then finally, viewing the results. After this, just save your project. So, that in future if you want to refer the same circuit you can it will be available in the server

(Refer Slide Time: 05:21)



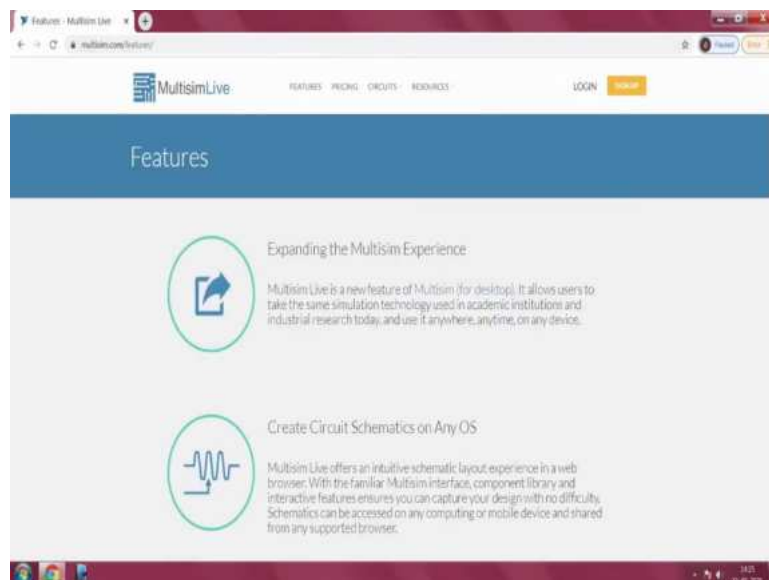
So, it also have multiple options just in order to explore or share the circuit you can collaborate with the community so that you can, discuss in the groups just by creating groups.

(Refer Slide Time: 05:33)



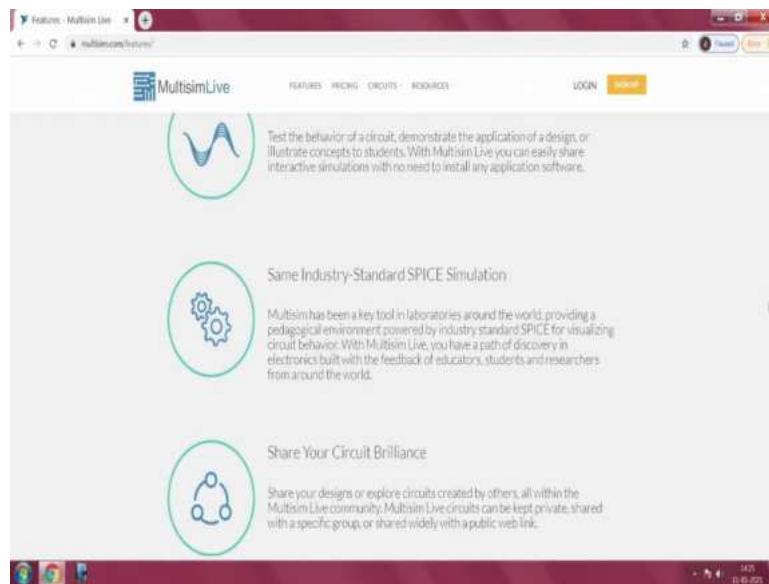
And then you can add circuit to your group and you can even share the circuits to people. So, since being an online it is more user friendly for the user to share and discuss in the online forums using software. So, for more about this working with the multisim can be referred using multisim website.

(Refer Slide Time: 05:59)



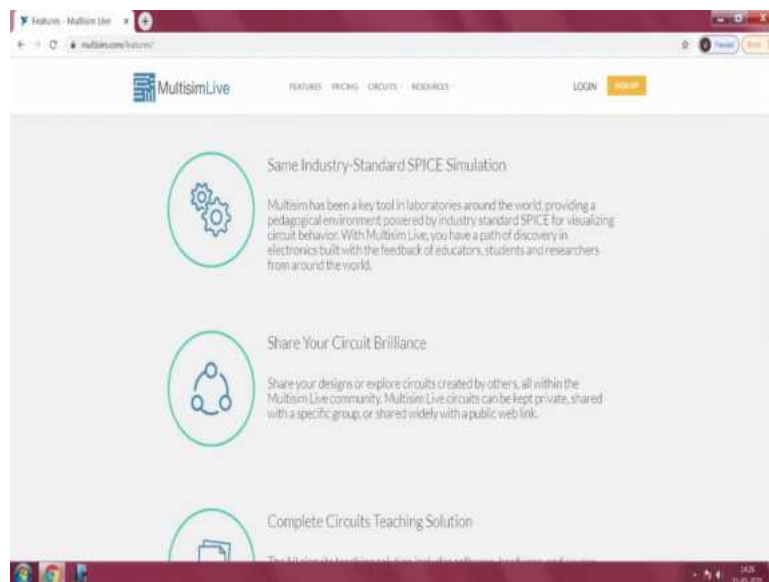
There are there are so much of data available in the internet to understand how to use this interactive simulation software.

(Refer Slide Time: 06:03)



So, one advantage is you do not have to require any installation file in order to work with this simulation software anywhere, any place if you have an internet availability, you can just log in into your account then you can start using the using you can analysing your circuit as well as whatever the circuits have already previously built.

(Refer Slide Time: 06:32)



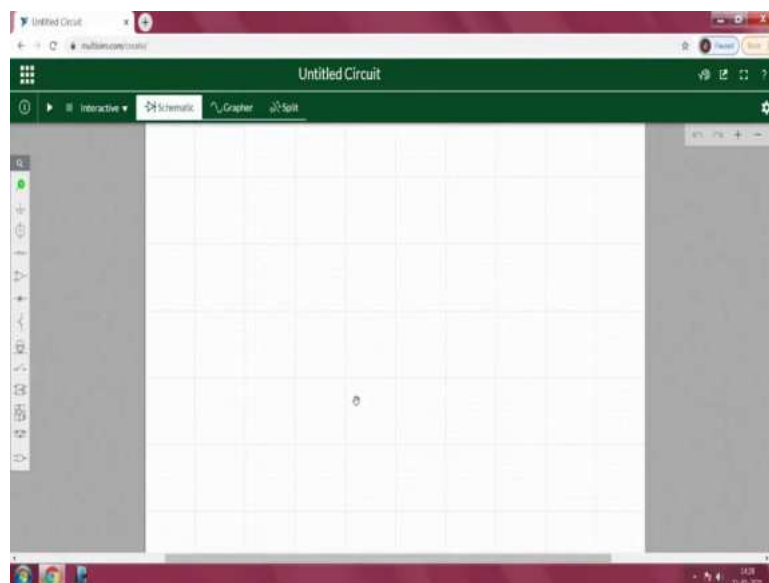
But you cannot use all the components available in the today's market. But however, you can analyse the behaviour of the logic in case if you want to constrain yourself to a particular component, then you have to better go with installation software where you will have a lot of database available for each component. So, we will start with building a small circuit just I am using my login account.

(Refer Slide Time: 07:12)



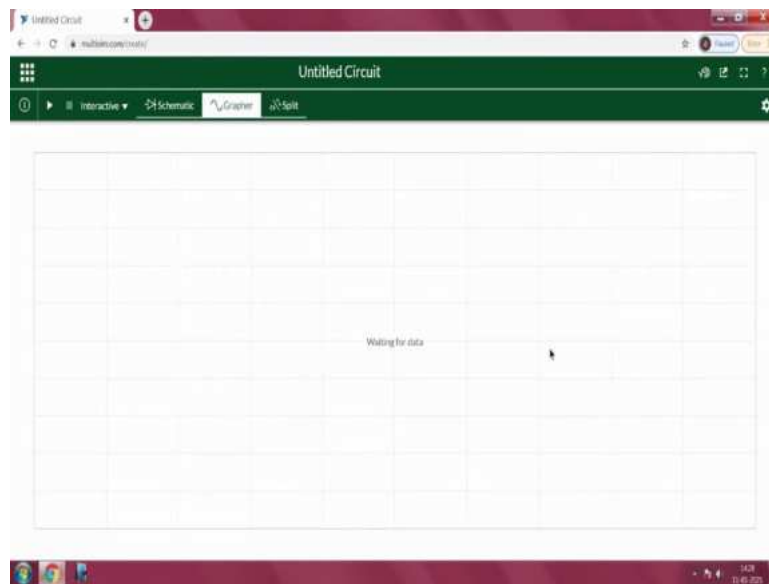
So, earlier lectures I have already recorded multiple circuits all whatever the circuits have created everything we can see here. So, since our idea was to create a new circuit, I am just clicking on create circuit which is available on the top right corner.

(Refer Slide Time: 07:36)



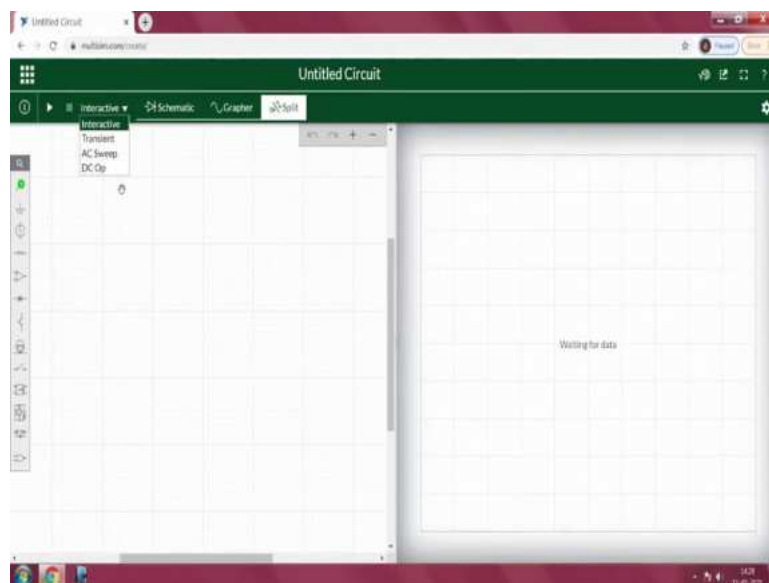
So, after clicking the create circuit as we have already discussed previously. A new page will be open where you can see the schematic window this is the schematic window the white colour completely. So, what are the circuit that what are the components available in the circuit has to be placed onto this schematic window. So, you have on the top side you can see 3 different modes. One is a schematic where you will be visualised with the schematic part.

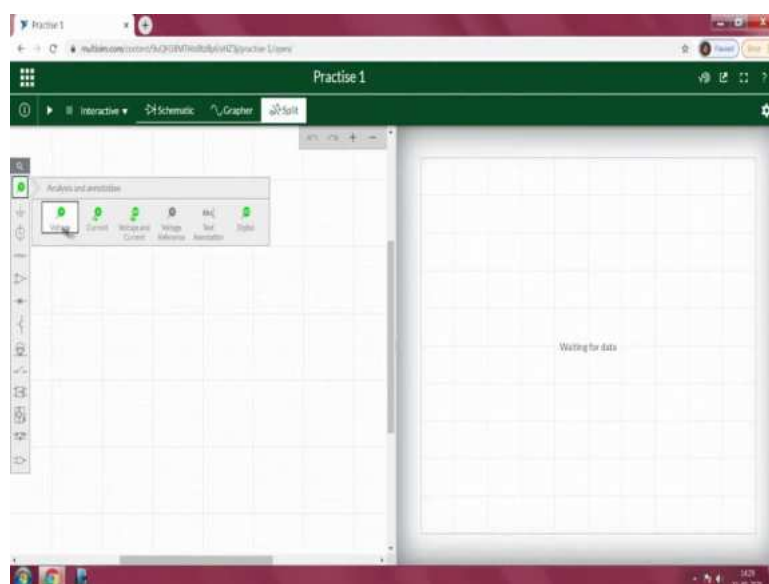
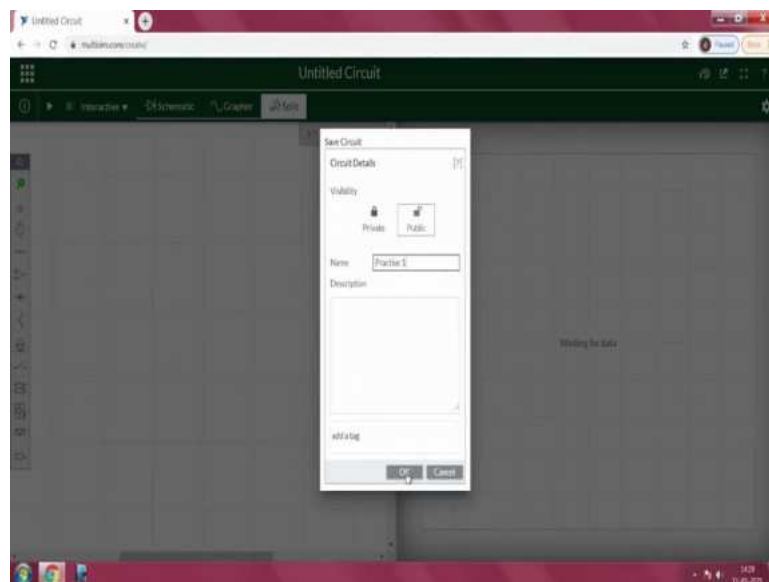
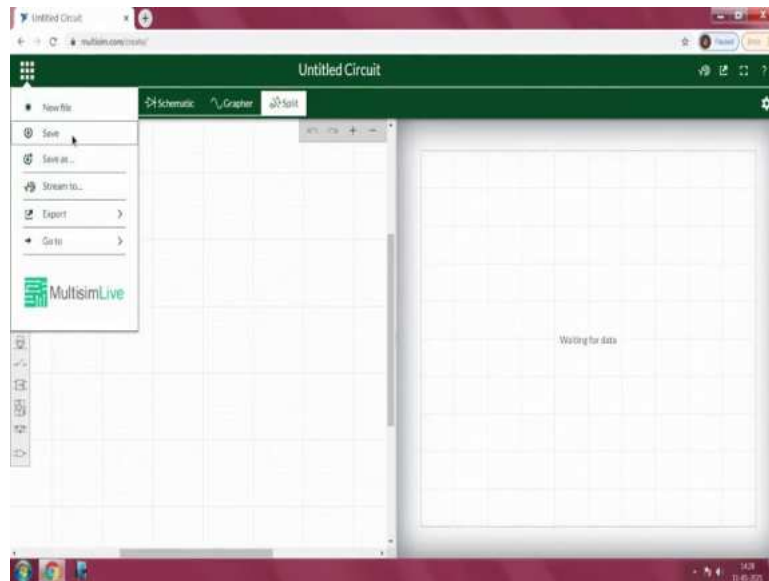
(Refer Slide Time: 08:11)

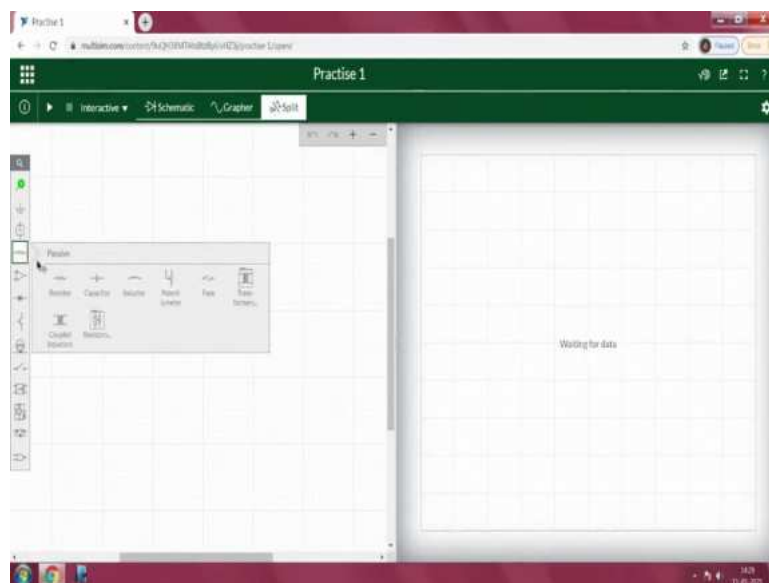
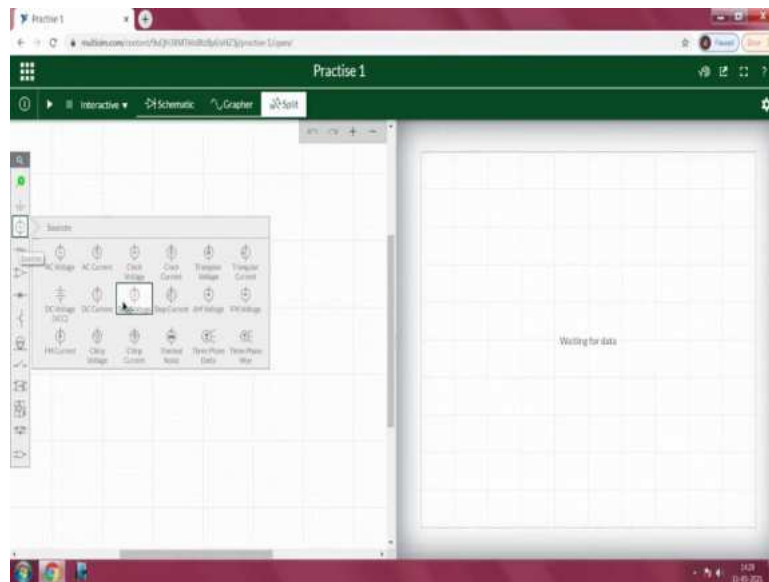
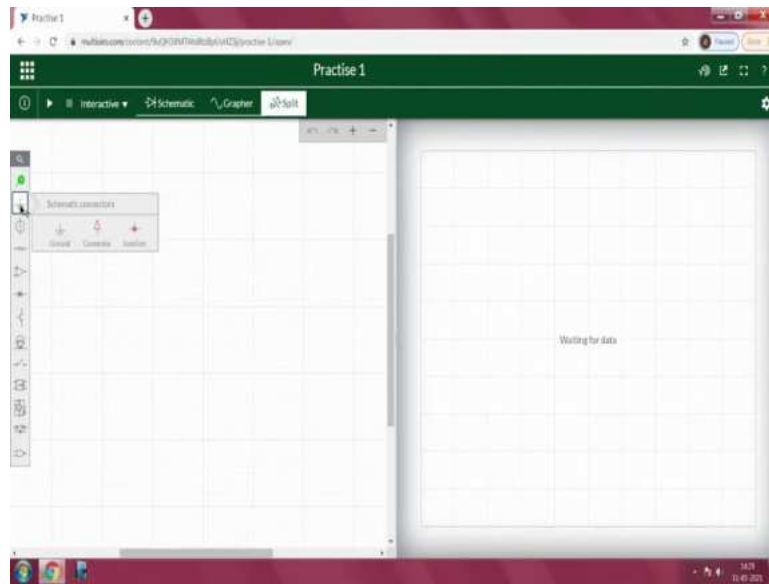


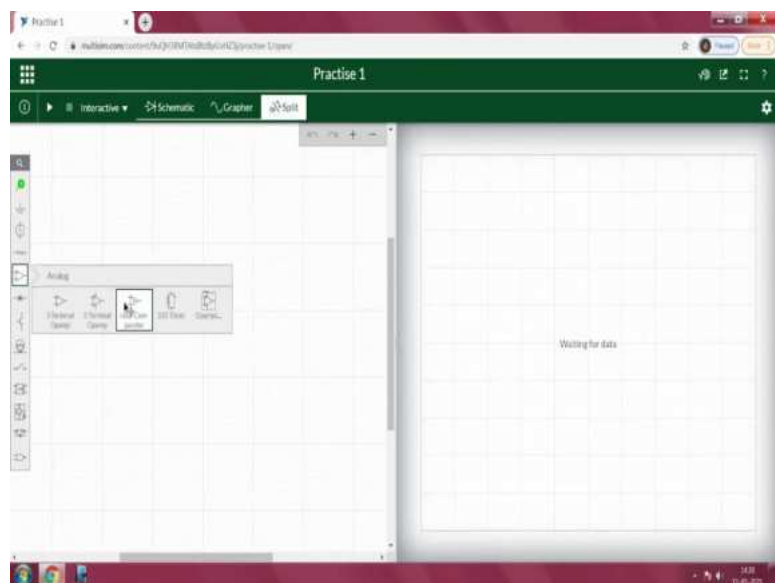
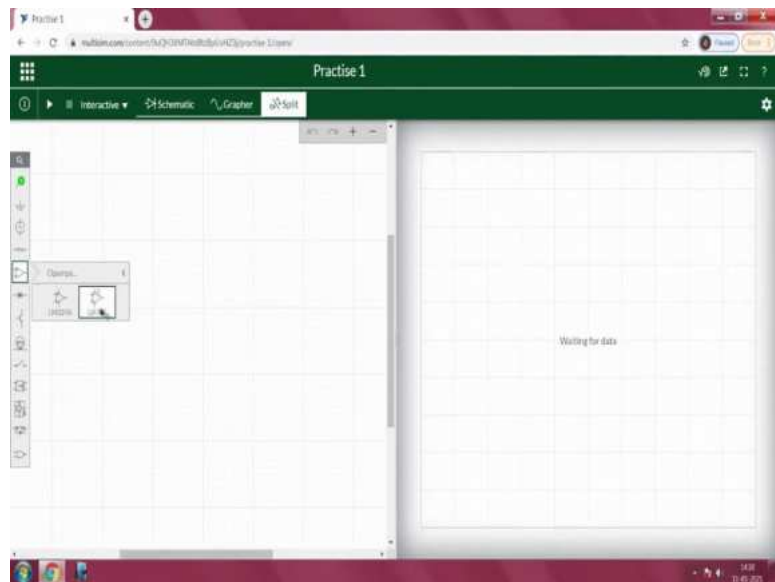
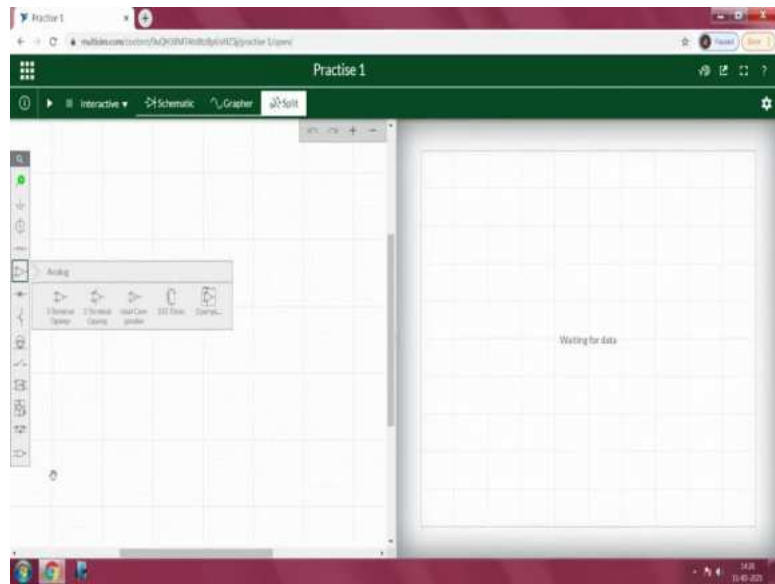
Then other one is a grapher. In order to analyse your data, output data as well as an input data with respect to the time you can completely analyse. It is similar to the oscilloscope used for the analysis.

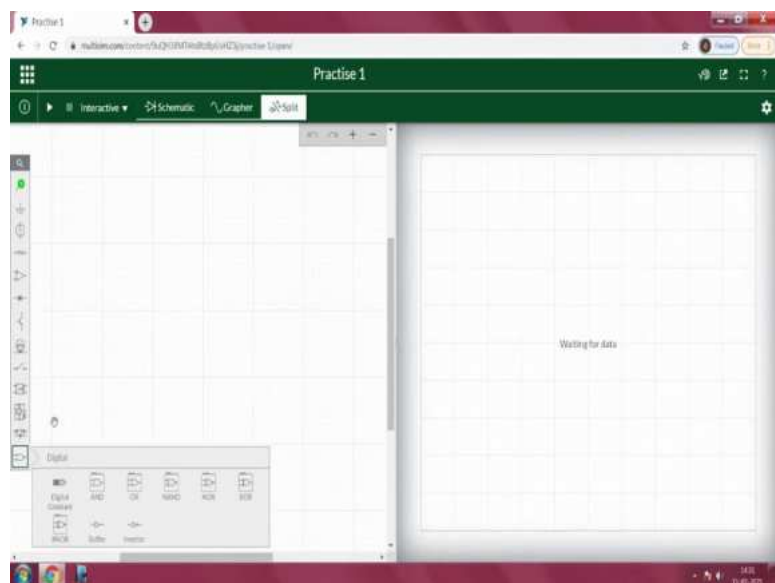
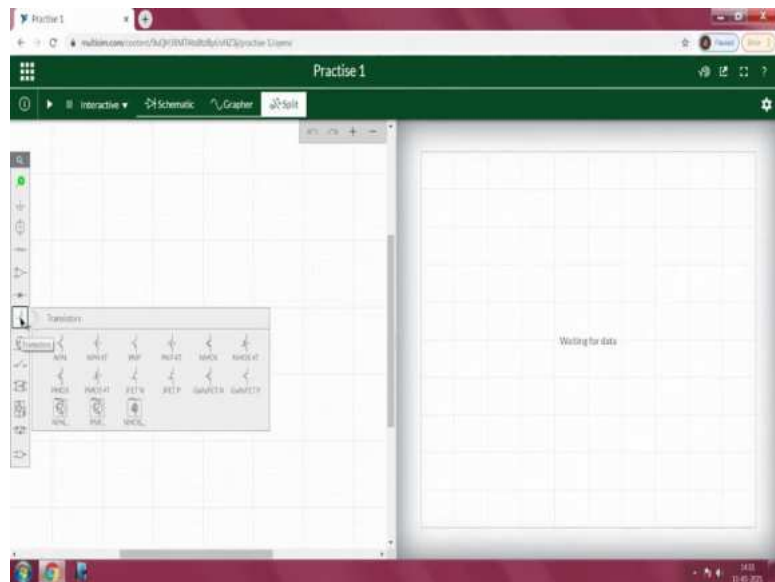
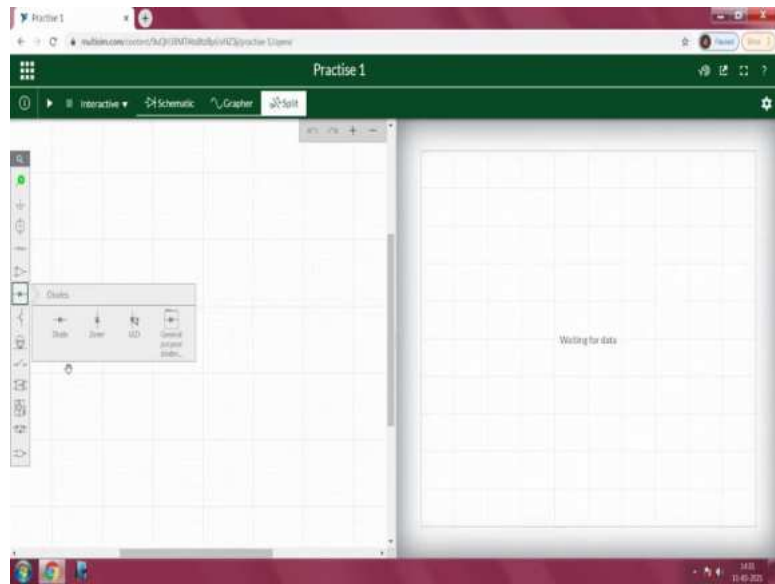
(Refer Slide Time: 08:29)











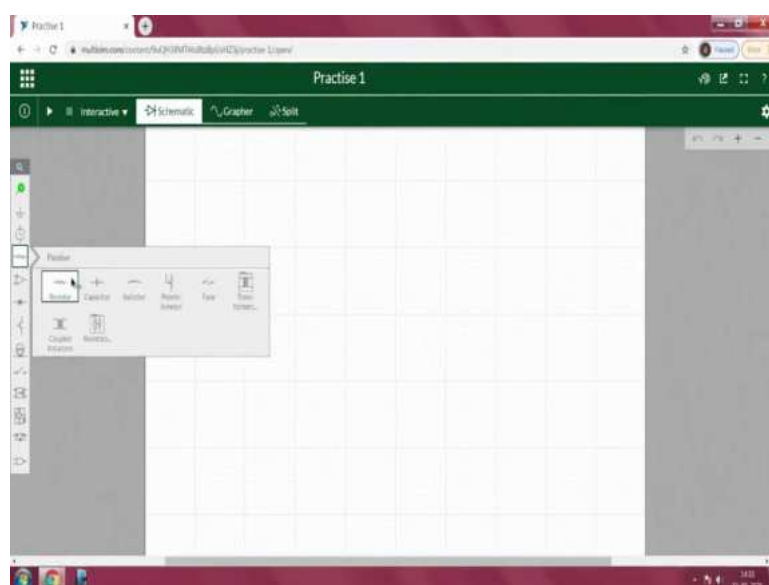
If you want you can split the window both your schematic as well as output data can be visualised parallelly. So, as I already discussed you have a different different settings for your simulation to process interactive, transient AC, DC. Whenever it is required for the circuit then I will be explaining about each and every simulation parameter here. So first let me save the file so I am making it as a practice 1.

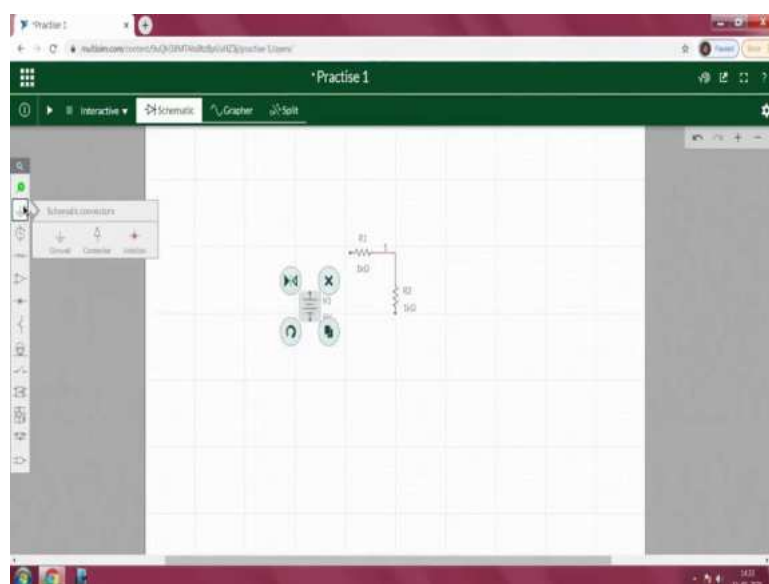
So, on the left side you can see different blocks. So, first one is probe block analysis and annotation where you will have a probes for voltages, current, voltage, everything. Then you have schematic connectors for grounding junction everything. Then you have a sources you have AC sources, AC voltages, DC voltages, step triangular, everything. Then you have a passive window where all the passive components like resistors, capacitors, inductors, potentiometers, transformers, everything will be available in this passive window.

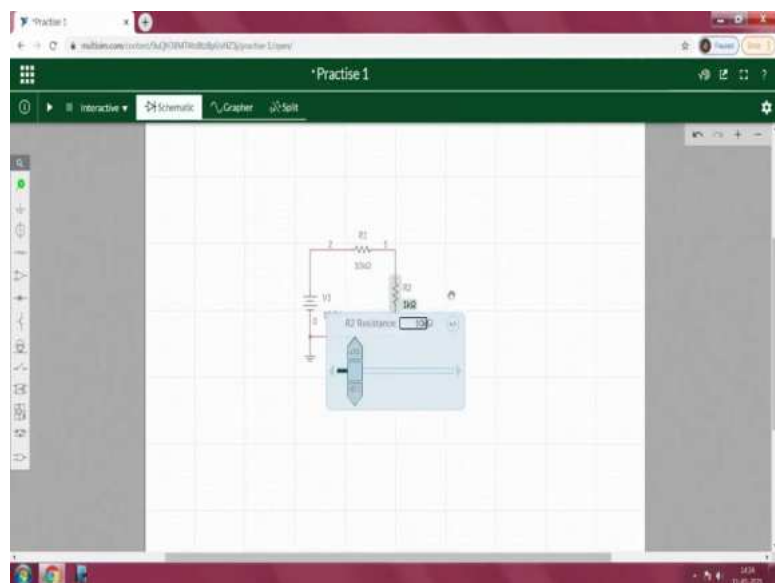
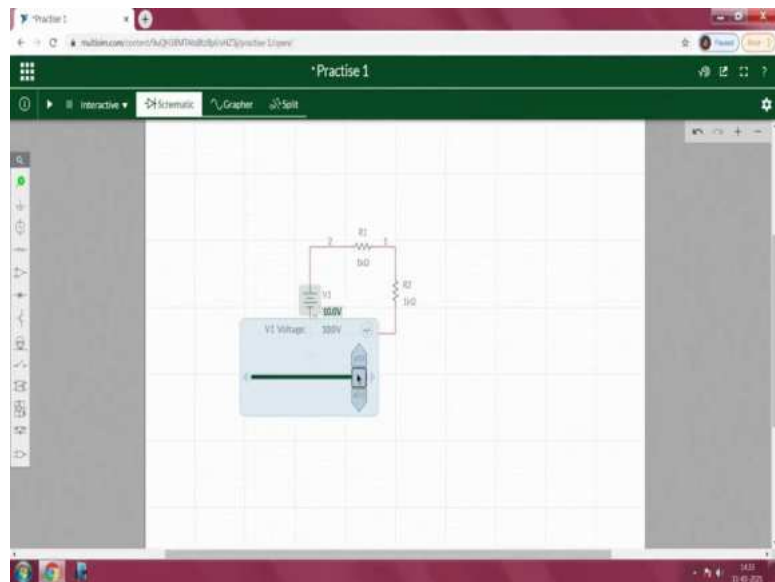
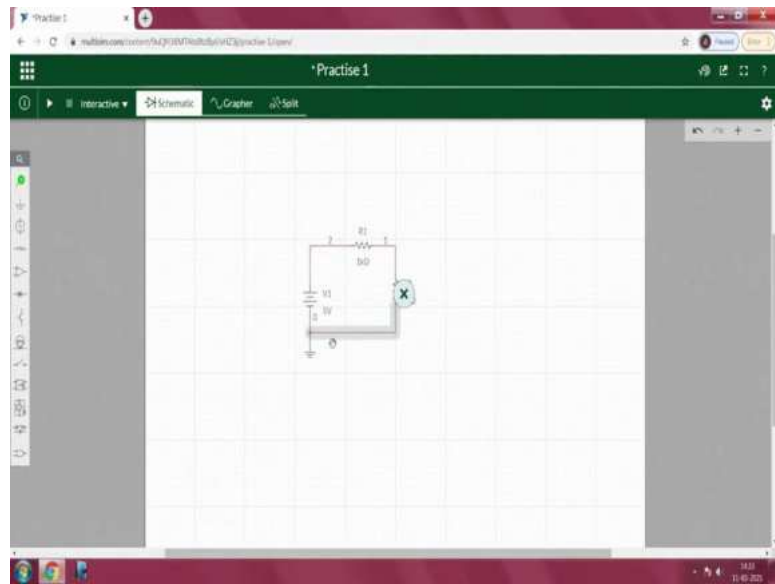
You have analogue blocks like Op Amp basic, analogue blocks, 3 terminal, 5 terminal, comparators, triple five. Along with that you have a generic IC of LM 741 as well as three to four. If you want to realise with the another IC like KL 084, 74, 72, 82 those cannot be realised using the online multisim. So, it does not have a complete database available with respect to the all the ICs.

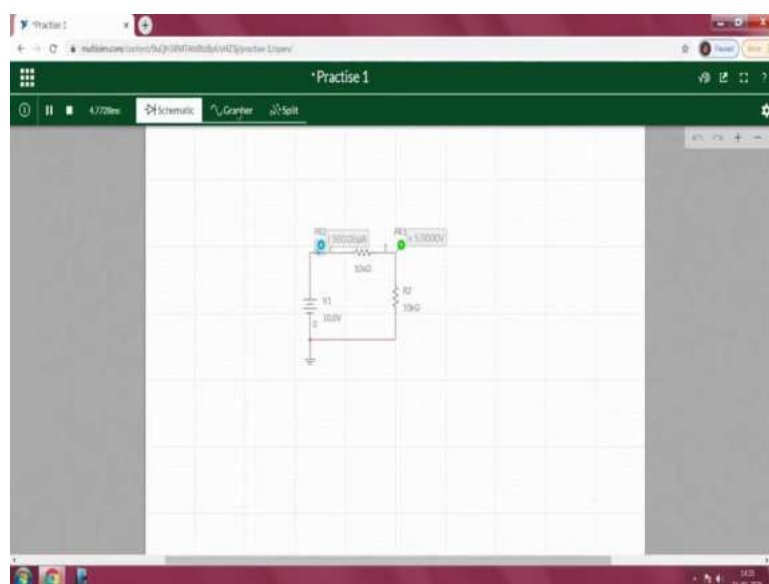
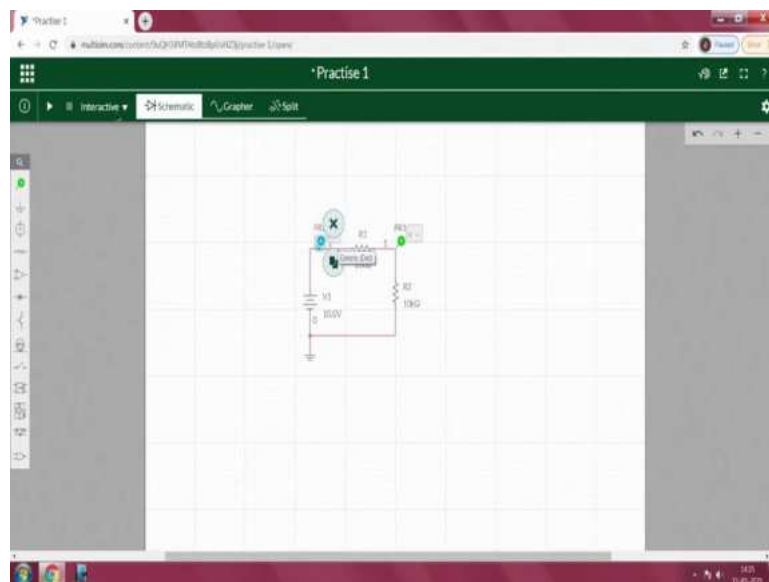
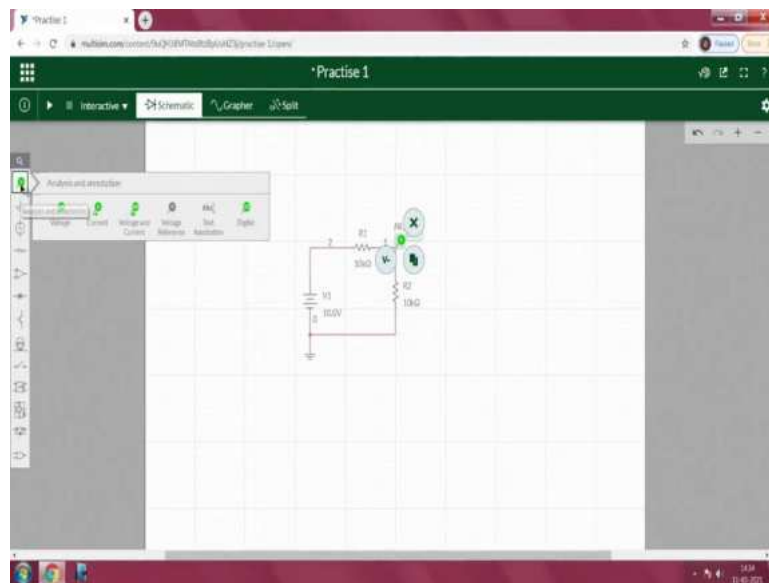
However, there is a software available multisim software available offline where you can install and you can work with the database. Then you also have a place for diodes to where you can drag and drop diodes required for your application. Then transistors and digital blocks.

(Refer Slide Time: 11:15)









Most of the components that is required for our applications are available here for analysis purpose. So, in order to explain you teach you the first component I would like to realise resistive divider network. So, for a resistive divider network we require to place 2 resistors. So, I will be taking the resistor from the passive palette.

So, I am going to duplicate the same thing by copying and duplicating it. After placing it when you take your cursor to the end of the terminal, the cursor will automatically change from the hand tool to your wiring tool. So, indicating that now this is this can be used for connecting between the two resistors. So, I will just click here so that it will start wiring. Then wherever it has to be ended just click at that place.

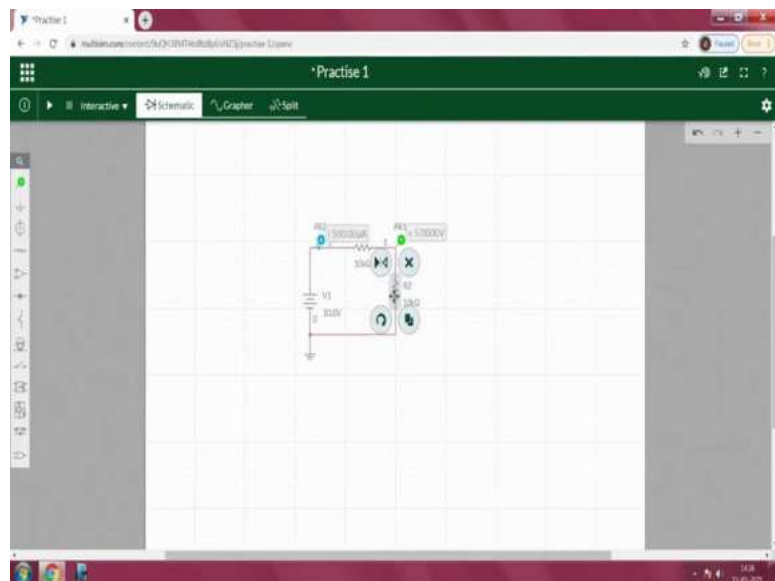
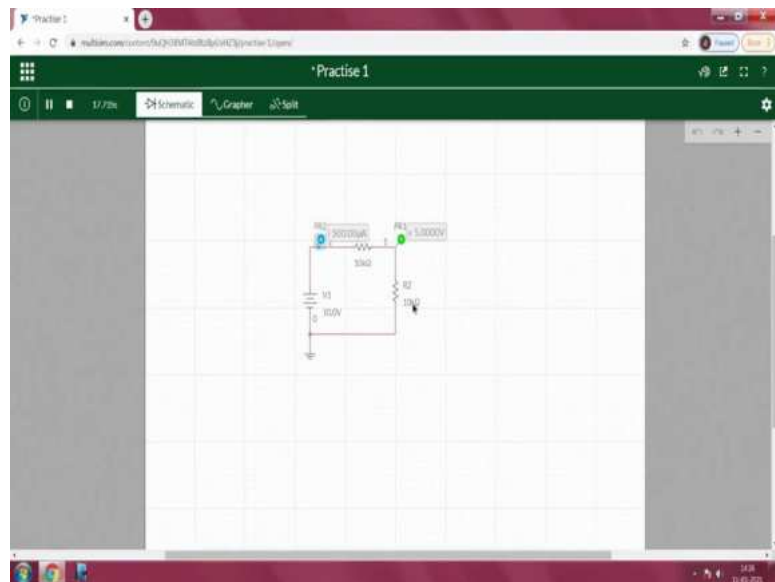
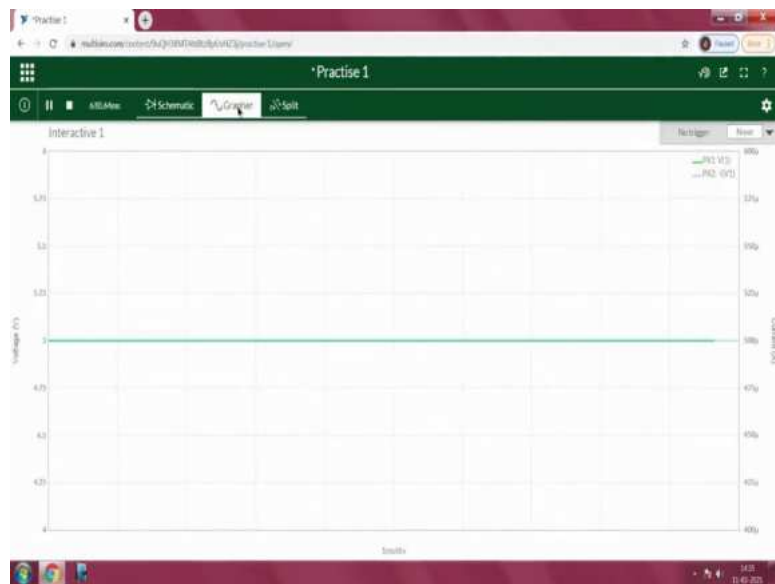
So that a wire will be connected between the two connections. In case if you want to delete just press on the into button I want to rotate it the R2 and just drag and place little bit down. Now, it is a resistive divider. So, in order to understand the working of it, after placing the component the wiring has to be done along with these 2 components you also have to place DC voltage source as well as ground.

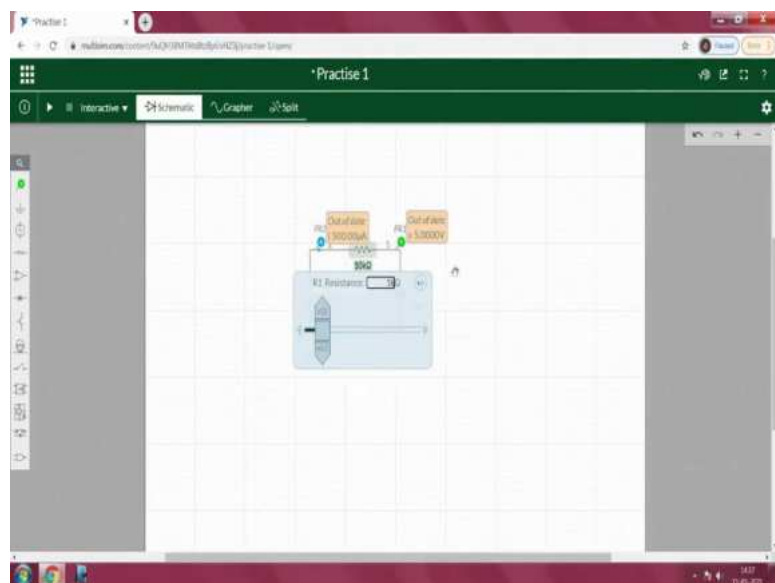
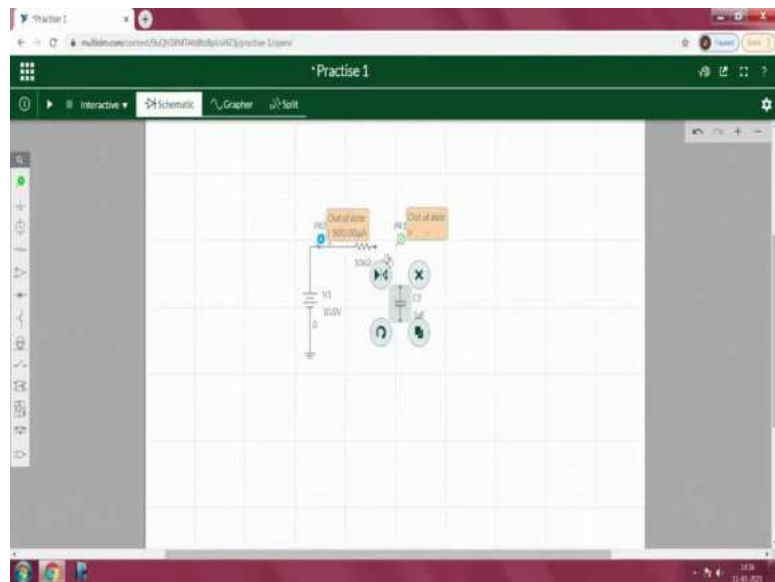
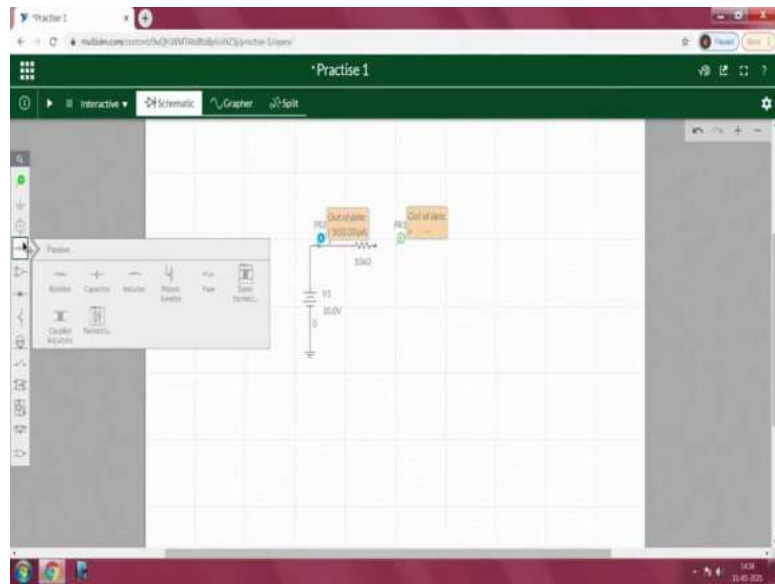
So, I will simply take DC voltage which is available in the source select DC voltage, then ground; ground available in this schematic and at this second pallet just placing it and making a wiring between the terminals wait slightly more down and then the terminal of R2 will be connected to the ground. So, after wiring it the next step is changing the values in case if you require to change from the default values I would like to change the voltage to 10 volts.

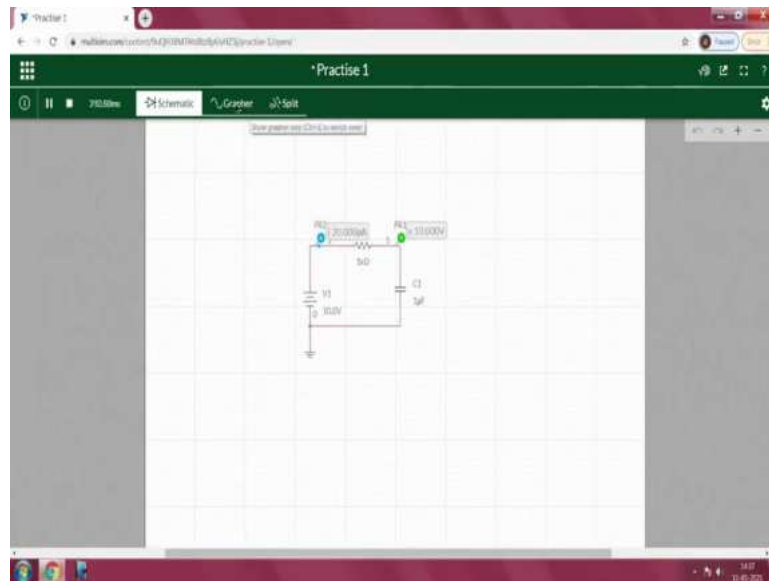
Then this resistance to be of 10k so even this resistance I am selecting it as 10k. Finally, to visualise the response, I can use a probe, voltage probe. I can simply replace it if you want to even understand the current flowing through the circuit along with the voltage you can also use the current probe. So here you can see the arrow indication of an arrow. So that means that if the current is flowing from the positive terminal to R1 then it is in the same phase so it will be shown with the positive.

If the current is flowing in either direction. Since the arrows towards right side and if the current is flowing in opposite direction it will be indicated with the negative, please do not confuse. Finally, I will be going with interactive session just an interactive then press run.

(Refer Slide Time: 15:02)





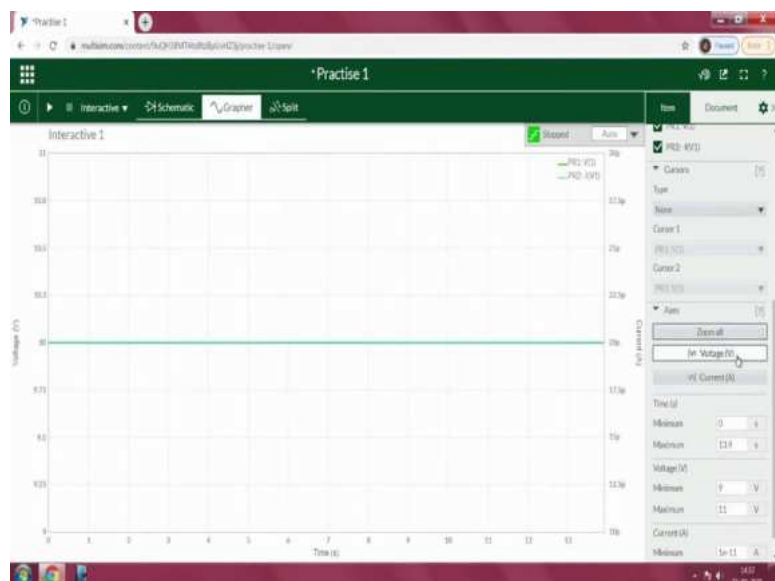
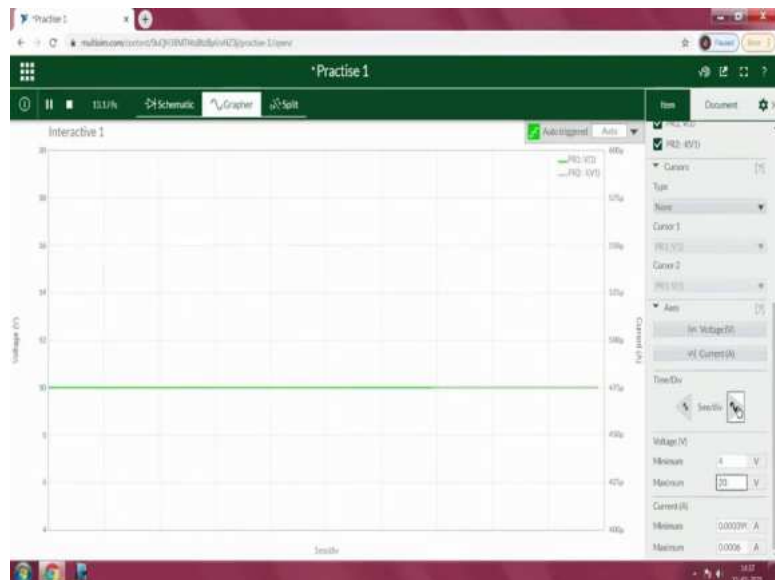
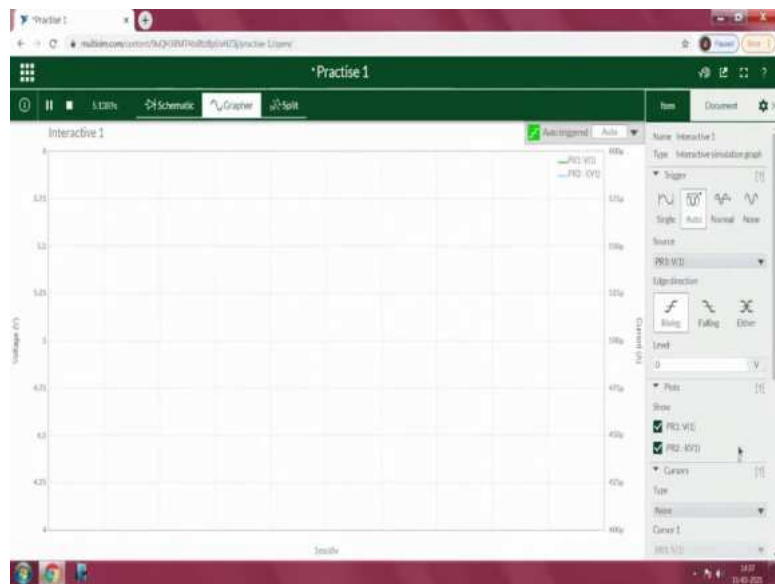


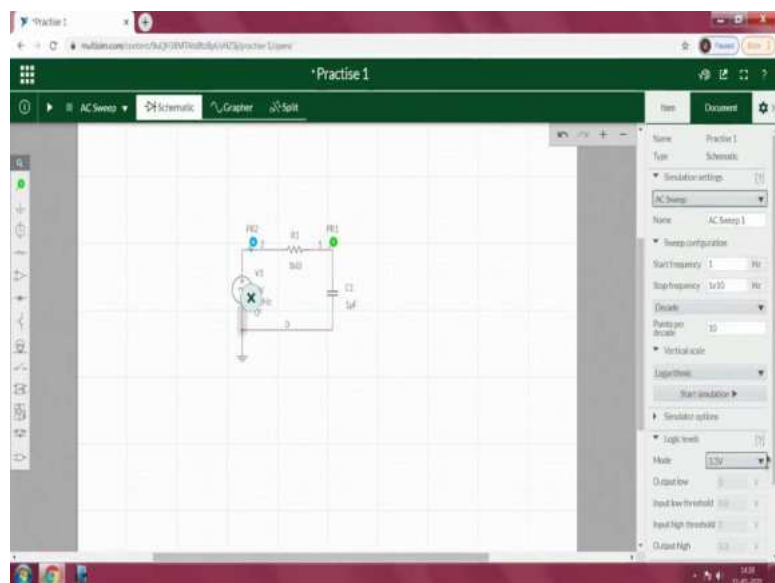
So, I will go to the grapher so here you can see since it is a simple resistive divider network and I am applying input voltage of 10 and the both the resistance of 10 both the resistances of 10k and 10k as we know that it is $\frac{V}{2}$ so since $V = 10$ so output will be 5 volts and both the both the resistance are in series so the total resistance is of 20 kilo ohms so 10 volts divided by 20 kilo which is of 0.5 milliamps of current.

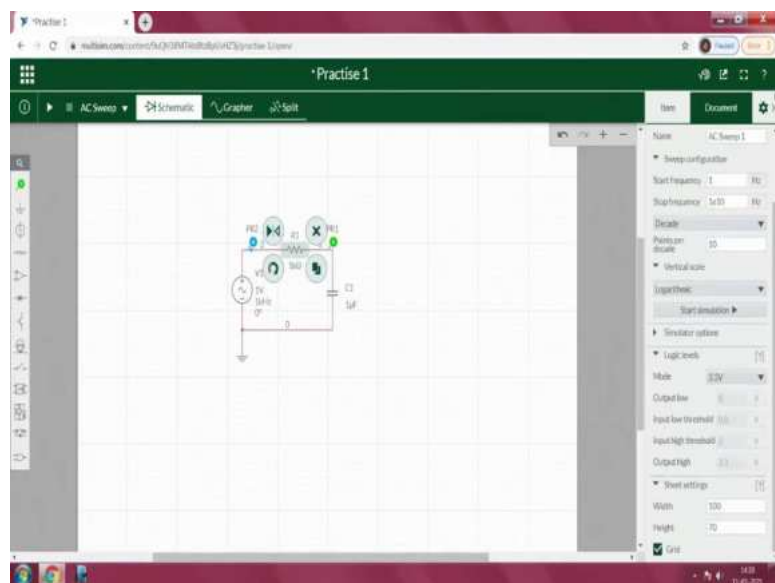
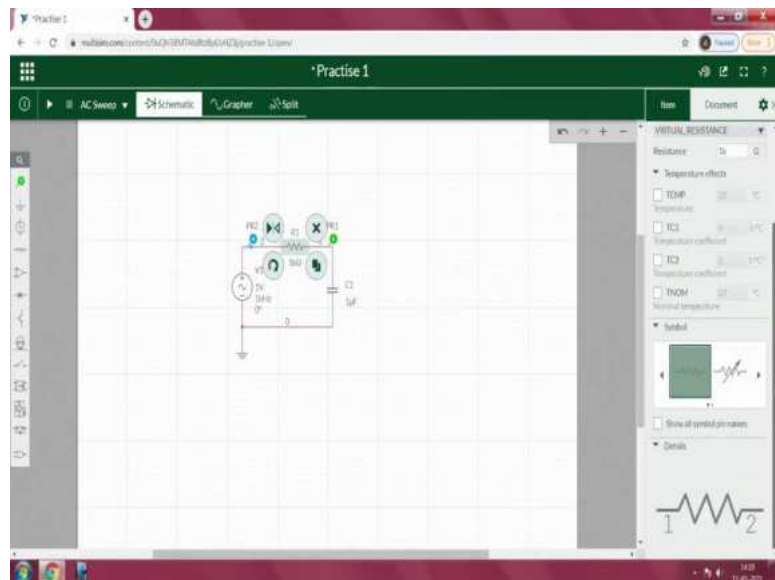
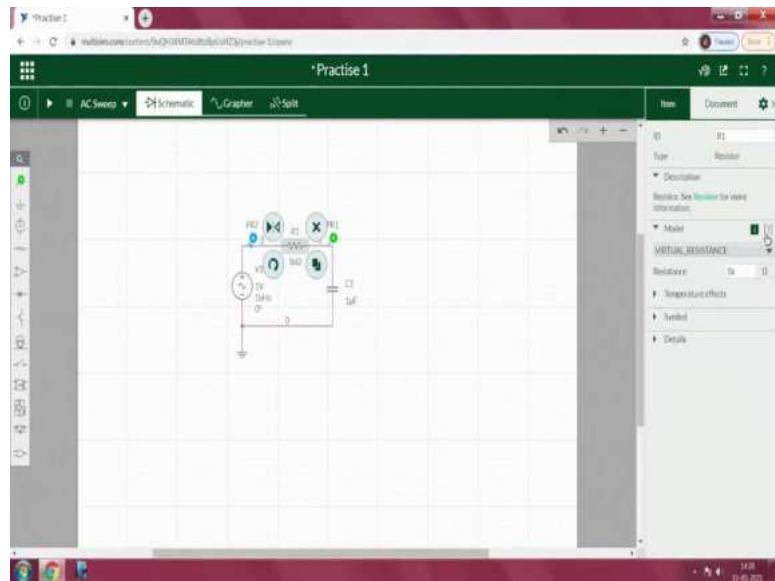
So, here we can see the 500 micro amps which is which is nothing but 0.5 milliamps. So, it is matching with respect to our theoretical verification. However, right now we can only see you know output voltage with respect to the time. Since, the input is a DC the output is also a DC and it is a constant since we are not changing any voltage. We are not changing any parameter either input voltage as well as resistance.

So, now I am planning to realise a simple RC circuit so that we can use some other mode of understanding for the simulation. So, rather than taking R2 as 10 kilo ohms I will simply delete R2. I will replace R2 with a capacitor. So, it act as a simple RC circuit, charging circuit, integrating circuit or charging circuit. So, 1 micro farad and 1 kilo ohms I am taking.

(Refer Slide Time: 16:51)





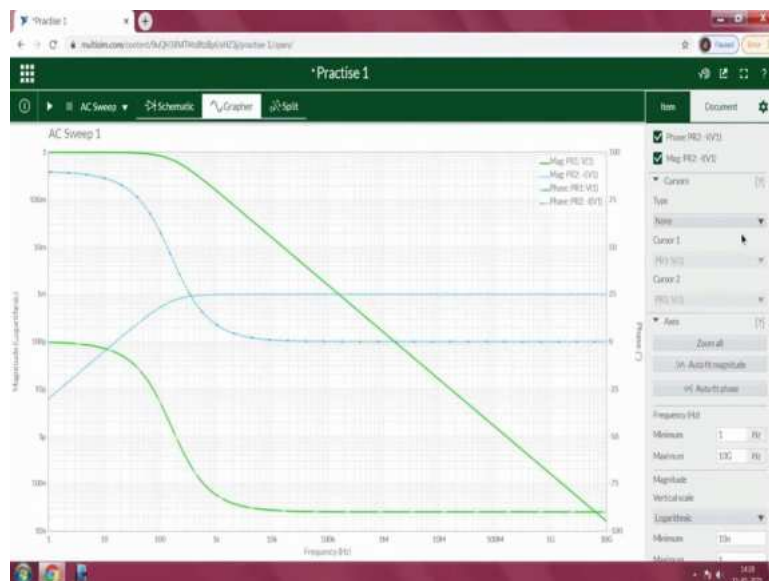


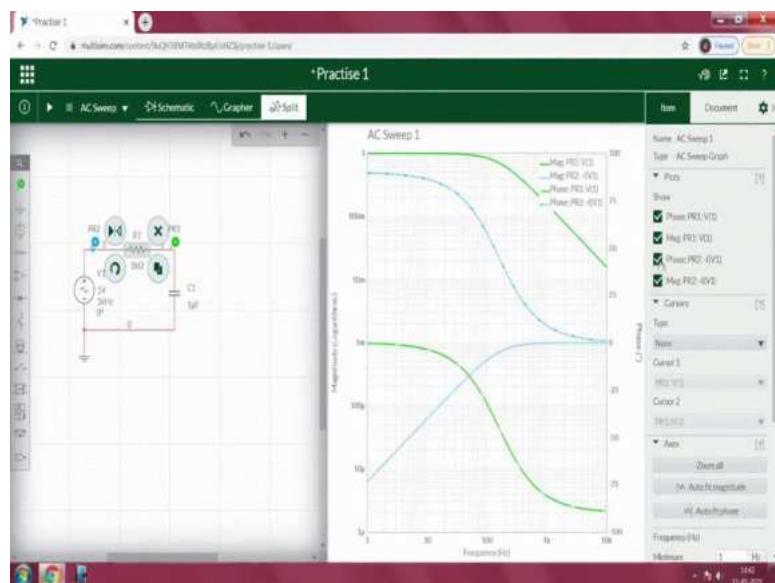
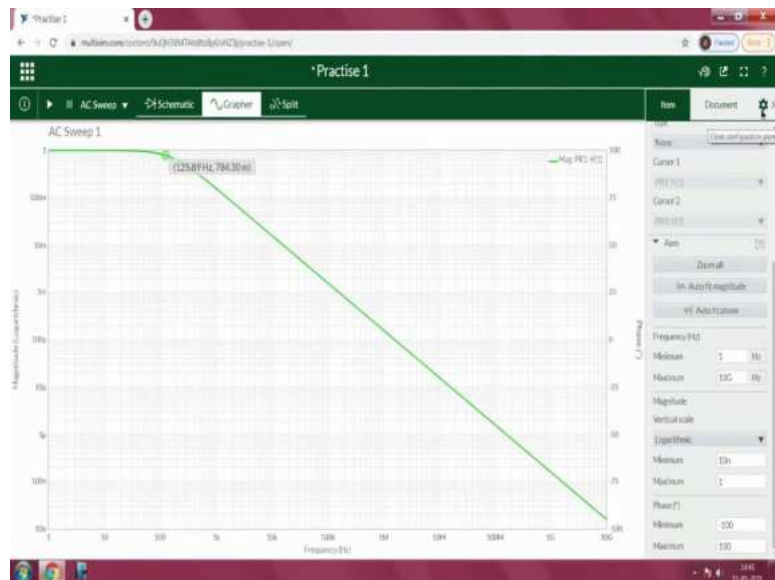
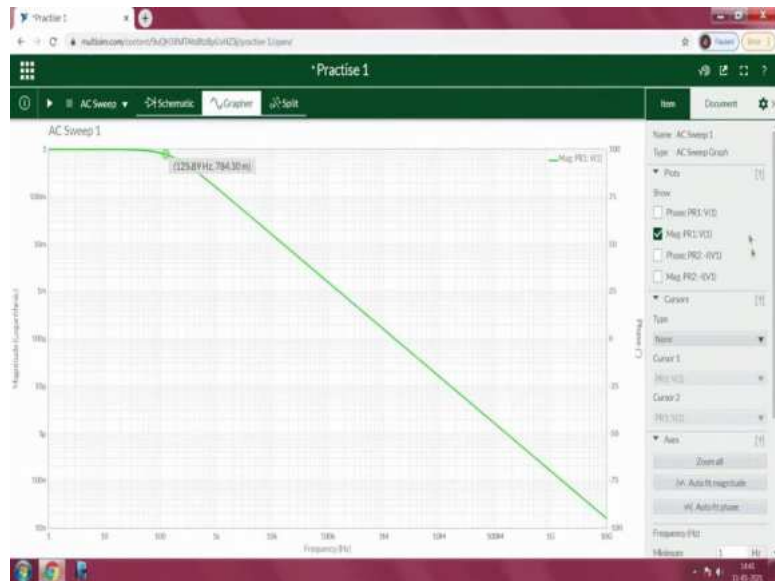
So, just running it graph, zoom on. So, as we know how RC circuit works initially at pump current so that it starts charging once it reaches to the maximum voltage the voltage across the capacitor will be always constant and it will be equal to your input voltage that you are that we know about how the working of in simple RC circuit. But instead of realising the initial phase of the complete circuit using interactive mode, I just want to go with a simple AC sweep.

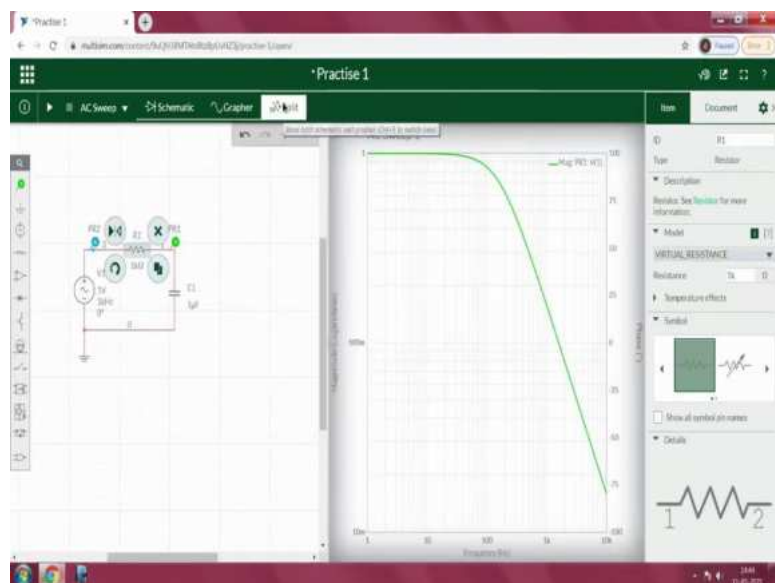
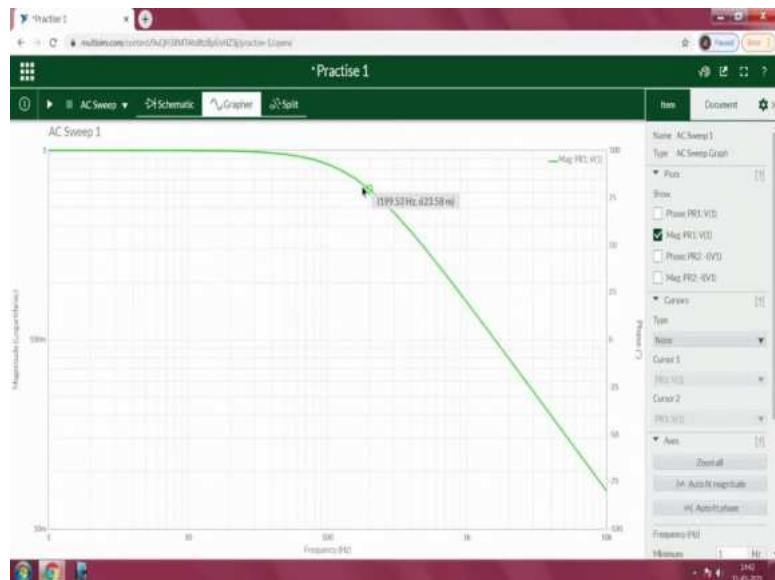
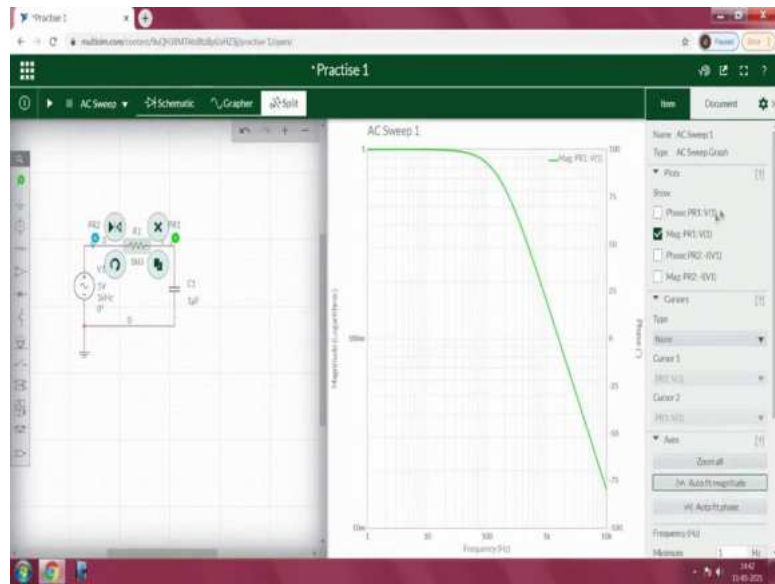
So, this I will be replacing with a simple AC circuit, AC voltage so that I can change different frequencies and see how exactly it is going to work. Now you can understand the filtering concept of RC circuit. So, I will go to the settings of AC sweep I want to start the frequency from one hertz to 1×10^{10} hertz.

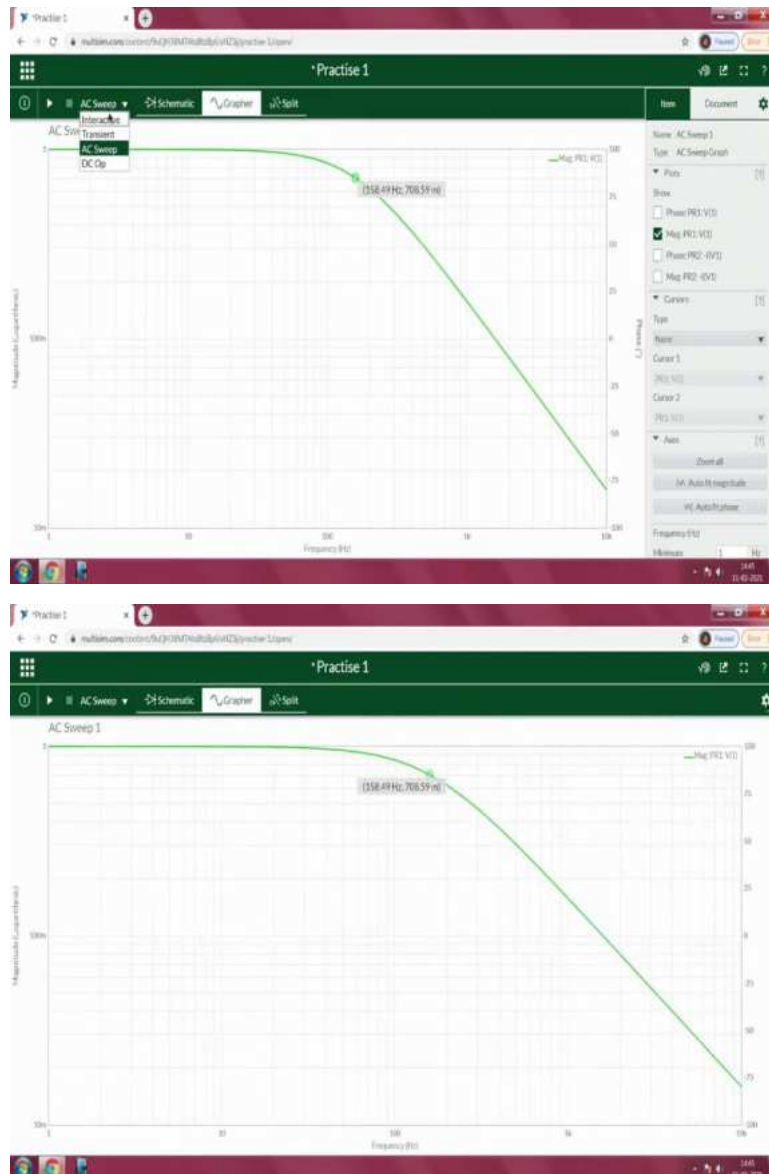
All the parameters looks good in case if any parameters to be changed here, you can simply go to the settings which is available here, then automatically the settings of that particular component will be available in case if even for the resistance R1. You can see here the complete parameters of, the model can be visualised here with respect to the symbols and the pin numbers everything.

(Refer Slide Time: 19:07)









Just run, go to the grapher. So here we can see the border plot. So here what you can understand so since I do not need the magnitude and phase for time being, I am not looking into the magnitude and phase for the input voltage. So here we have created a current as well as PR1 which is a voltage probe. I do not want to see the current information here. So I will disable PR2. In order to disable the PR2, simply go to the settings window of the grapher.

Here we can see the curves and the plots which to be visible on the screen. So, I am just de-selecting the magnitude and the phase of PA1 and I am just plotting only for the magnitude of the PR1 alone. Disabling the PR2 graphs and enable only the magnitude of PR1. So, here we can clearly see that so, after a certain frequency it stops allowing any frequency which is filtering output and this is a simple low pass filter.

So, you the border plot also looks like the low low pass filter that means, it will allow the frequencies only particular value after that it will not so, in order to understand the cut-off frequency you can plot the 3dB. So, since it is 1, 3dB lesser than that will be the value so, that the cut-off frequency will be close to, so, this is one so, this will be 700 approximately. So, it is it is of around 125.89 hertz is the cut-off frequency.

If you want to understand if you want to see the cut-off frequency $\frac{1}{2\pi RC}$ just analyse that and match with respect to the cut-off frequency. Any parameters to be changed for example, so you do not want to change you do not want to see the frequency plot from one to 10 Giga rather than that it is okay if we can plot up to 10 kilo so in order to change the parameters, just go to the settings window here.

But you have to go to the sweep settings window. So, just go to the schematic then it will automatically goes to the sweep settings window. Then here instead of going with 1 Giga, I am going with just 10k then I am splitting the window then plotting it. So, as I already mentioned just zoom all everything, then magnitude and the phase plots of PR2 is disabled only the magnitude plot has been enabled.

So, auto fit magnitude. Then, for better visualisation, I am going to the grapher here we can clearly visualise so it is roughly around between 199 and 158. So, when we calculate the cut-off frequency, we got around 159 hertz. So given from the analysis we can clearly see the cut-off frequency of this is close to yes, this is 1dB, 2dB and 3dB, 158.49. We got 159 this is a theoretical versus practical.

So, I hope this is clear for you to understand. How can we use an online multi circuit simulation software in order to realising the basic and for analysing of circuits? With this I will stop this thing. Any doubts we can you can please contact in the forum. Thank you.