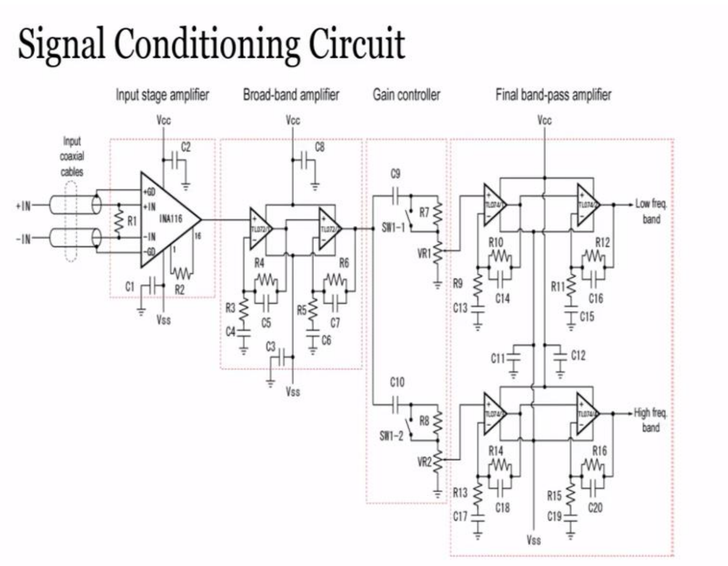
**Introductory Neuroscience and Neuro - Instrumentation**
**Indian Institute of Science, Bengaluru**
**Lecture No. 30**
**Signal Conditioning Circuit for EEG Bioamplifiers**

Welcome to the module. So, today, we are going to see the implementation of the signal conditioning circuit which the professor has discussed in your previous class. So, the idea of the circuit is to do the signal conditioning of the acquired EEG signal. So, as you have already seen about the circuit, we have different stages of signal conditioning that we are doing it. The first stage is for the first stage is called the Input Stage Amplifier, which uses an instrumentation amplifier. The instrumentation amplifier that we have selected here is INA 116 this is because it has provided a very high input impedance.

(Refer Slide Time: 00:58)




Very close to one somewhere around 1 peta ohms and it also has a very low bias current. Not only that, the features the feature of this is INA 116 is that it has provided with driven shield input as well as guard pins. So, because of these, it holds the shield of the input coaxial cable at the same voltage as the electrode connected to the input through the guard drive pins. So, what will the advantage of it? So, because of this the capacitance between the electrodes and the shield is canceled, thus preventing electrostatic interference.

So, that means, it can provide capacitive coupling between the input pins, as a result of any noise since, this is an instrumentation amplifier where we will be always providing the differential signal at input IN plus and IN minus. So, since it always does the difference between these two, any common noise available at IN plus and IN minus will be automatically canceled, cancels out. So, because of these advantages with INA 116, the input stage has to also have a very high input impedance it has to choose and go with INA 116.

So, the next stage is to remove the other frequency of the noise signals available in the circuit are available in the recorded signal. So, we are interested in the frequency of somewhere around 1 hertz to 3.7 kilohertz. So, that is why this broadband amplifier stays where it is a bandpass filter, a two-pole bandpass filter has been designed to go with 1 hertz to 3.7 kilohertz of frequency. The selection of R3 R4 C4 C5 R5 R6 and C6 C7 has been chosen in such a way that the frequency will be from 1 hertz to 3.7 kilohertz.

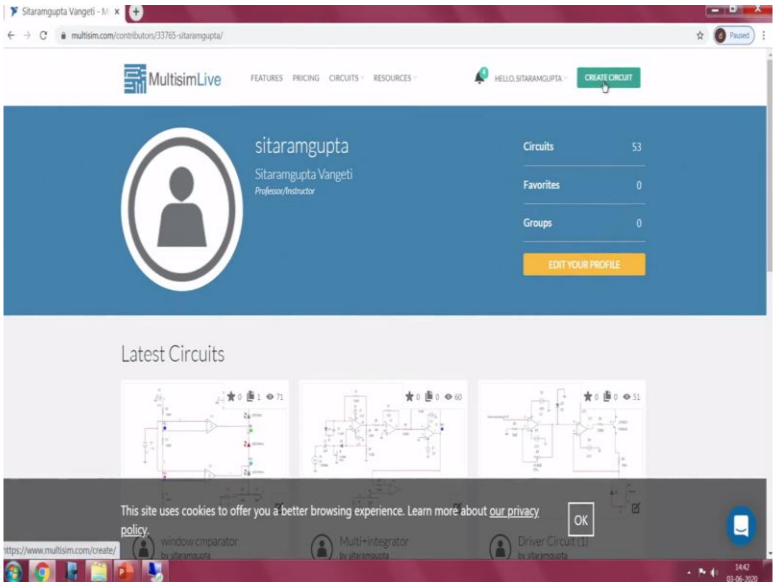
The second stage is again controller states where depends upon the signal amplitude, we can further attenuate the signal by using a switch 1 1 and switch 1 2 as a result in case if we required the gain to be attenuated this particular stage helps to attenuate. Whereas, the last stage where it is a final bandpass filter stays can you know divide the signal into two different bands, one the top part which will provide us the low-frequency band.

The frequency or the bandwidth of this particular low band frequency circuit is 1 hertz to 340 kilohertz above which it cannot transmit any signal and whereas, the bottom part is from 320 hertz to 3.4 kilohertz. So, as a result, whatever the signals that we have provided the input stays we can see the low-frequency band signal on this at this particular point, whereas, a high frequency of greater than 320 hertz up to 3.4 kilohertz signal can be viewed at this particular point.

Since our idea was to understand the working of these particular individual stages of signal conditioning circuit. Now, we will see how do we implement this circuit using some simulation software circuit design simulation software's. So, for this application, I will be I have been I am choosing a Multisim, an online Multisim where anyone of you having internet can be directly connected to the online Multisim just by registering it.

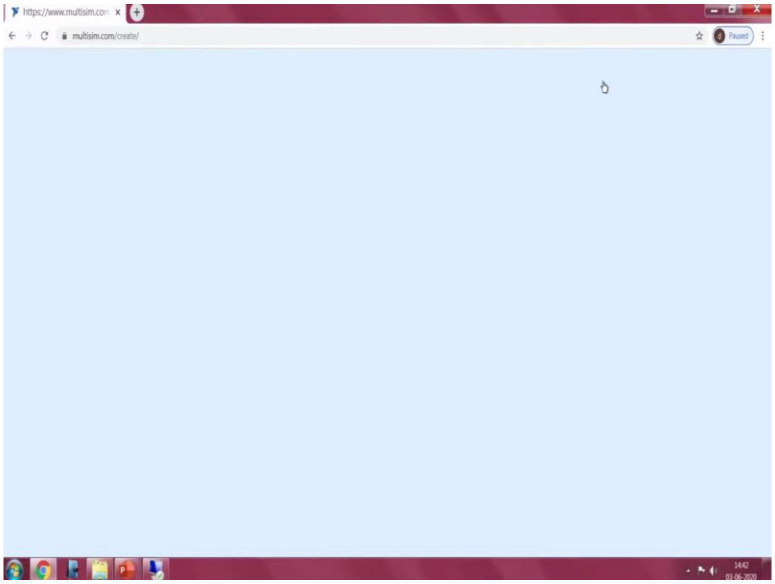
And you can implement you can visualize the functionality of the individual blocks you may not get the direct operation amplifiers or the direct part numbers in the online Multisim since it is an online version and it is open to everybody but you can simulate the functionality of the circuit part. So, now I will show you each one. So, I will start with the broadband amplifier stage. Since INA 116 is not available in the database of online Multisim. So, we will be implementing you know the instrumentation, instrumentation amplifier configuration using Three Op-Amp based instrumentation amplifier. Just to understand how exactly the working of instrumentation amplifier does. So, let me open online Multisim.

(Refer Slide Time: 05:33)



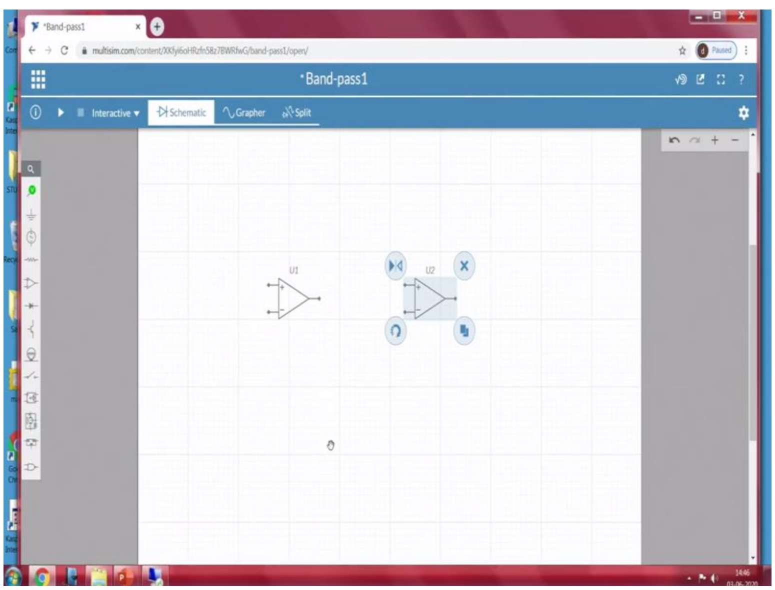

So you can open so, you can open online multisim using multisim.com. So, you can register it so that you will have your login ID and password. So, right now I am using my login ID and password. So, once you open the multisim on the right side top corner you can see the creates circuit.

(Refer Slide Time: 05:57)




Just click on it so, that an interface to create online multisim can be to build our own circuit can be seen. So, since we have already seen the introduction of the multisim in the previous video, now we will be directly going to the implementation of different stages here.

(Refer Slide Time: 06:13)



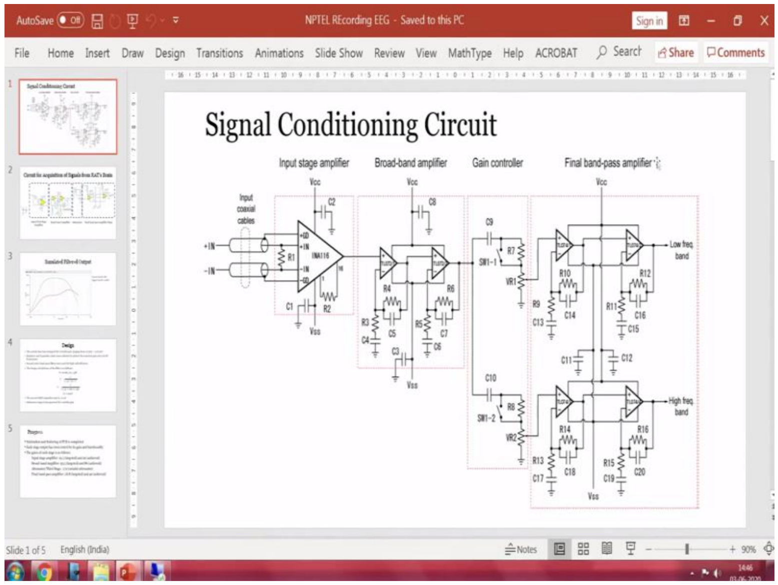

So, first I will try to save with the name Band Pass 1. So, in the circuit that we have seen, we have used a TLC 272. So TLC 272 has an a features like very high sleeve rate. But since this is an online version, we may not have to may not get all, all the database of the components available in the market. However, there are a generic blocks available at the same time very few online very few blocks available which supports the online which supports the availability of the components in the market.

Here, you can see the difference as we have already discussed, this is a circuit builder part where we will be dragging and dropping the required components for building the circuit. And here is a menu part where we can see whether we have to only work with the Schematic or if you want to see the Grapher. So, to analyze to visualize the outputs and inputs that we have given to the circuit are coming out from the circuit. And if you want to see both the Schematic and Grapher in a split window mode, we have a more there is a more called Split.

And there are different modes of operation, Interactive mode, Transient mode, DC sweep and DC operating. We will see depends upon the application and depends upon what kind of simulation that we are what kind of analysis that we want to go with for the circuit, we will select different modes to understand that to analyze the circuit part. So here, there are different menu options to select different blocks.

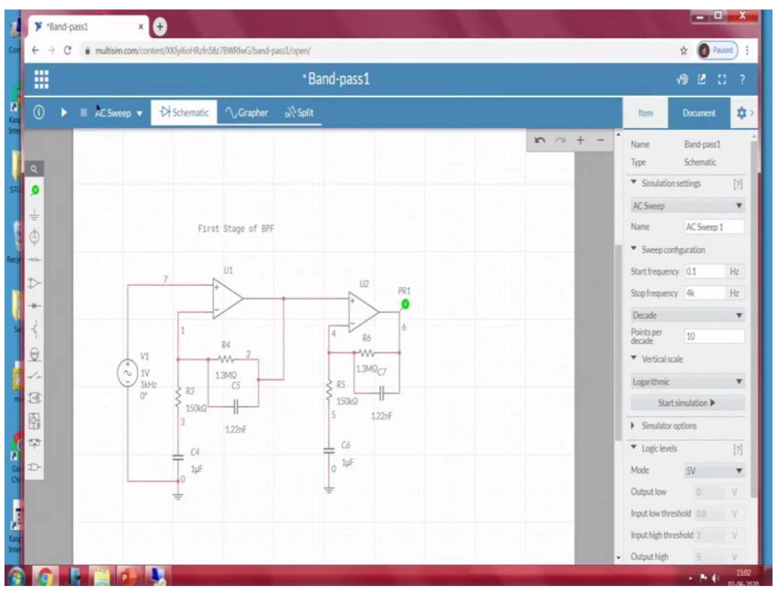
So, since we intended to build a bandpass filter, which is on the first stage, which we named it as which the name does broadband amplifier, I am selecting two operational amplifiers either I can go with generic operational amplifiers like 3 terminal op-amps, 5 terminal op-amps or op-amps with which are commercially available like LM324 or UA741 general-purpose operational amplifier. So, in this case, I will go with the 5 terminal Op Amp or you can go with simple 3 terminal Op Amp. So, we require a two operational amplifiers in this case.

(Refer Slide Time: 09:00)




And if you look into the circuit, so we have R3 R4, R5 R6 total 4 resistances, and 4 capacitance.

(Refer Slide Time: 09:10)




So, I will take four resistance from the passive component block. So one at this point, then one more here one more. So, I have added four resistance, I will change the resistance value. So, if you want to change the resistance value, just click on this particular resistance box or you can click on the number. Here we can see the pop-up window where it allows us to change different resistance value. Our another ways just select this, just go to the settings tab, which is there on the which is called as open configuration pane. Which is there on the top right side, Click on it here, we can edit the resistance number or ID of this particular passive component and, and its resistance value. In case if you want to further add some other effects like the temperature effect of the resistance, you can enable this click on this particular place and you can provide the temperature relevant to this. And you can further see what type of symbol it shows and its details to. So, these are the parameters that we are using for the simulation settings that we are using for this particular builder.

So similarly, we also have required passive components of capacitances. So, I will go with the again passive capacitor. So, one should be parallel. Another one should be in series with this resistance, then one more here, and one more here, so total 4.

So if we look into this bandpass filter, so this is a noninverting type where the input is directly connected to the positive terminal. So that is why this is a non inverting based. And so we will try to rename the same, same IDs of this resistance and capacitors we will call it as R3 R4, C4 C5, as well as we will go with R5 R6 C7 C6 to.

So, I am naming this as R3. So, I go from here. So, I name it as R6 and I name it as C7. This is the numbering that we have used in our circuit**.** This is R5 and whereas this is C6. So, this is C5. And this is R4 C4 R3, this is R3 and this is C4. So, now, we have to make the connections. So, when we look into the connections, the negative terminal, the negative terminal of an operational amplifier is connected with R3 and R4 l will do the similar connections here.

So, wiring is very simple, just when you place a cursor at that particular point at the terminals of capacitors or any passive devices, the cursor will automatically change from the hand tool, the wiring tool, this is called a wiring tool. So, just click on that then wherever you it has to be connected, you just place it if there is a place to create a junction, it will pop up like a red-colored circle as you can see here, just click on that.

So, that it creates a junction and it will be connected to that particular junction. Similarly, on the other end, I am doing the same way, just single clicks, then this part I will be connecting with C4 and the output has to be connected with this particular junction. I will change the resistance values to 150 kilo-ohms of R3 as per our required the filter frequencies. So, since this particular filter has been designed for the bandwidth of 1 hertz to 3.7 kilohertz. So, the values has been selected according to the design and C4 is 1 microfarad I am returning the same whereas C5 is 1.22 nano farad. And R4 is 1.3 mega.

So, other kind of C4 capacitor has to be connected to the ground. So, I will go to the schematic connectors, which is a second tab on the left side, where we can see different kinds of connectors, ground the connector junction. So, since we intended to connect to the ground, I select that I, you can just click on this ground to place it on the circuit builder page and click and you can place it on the page. So as usual, at the end of the connector, you just click then on the other end, you just click so automatically the wire will be connected between these two terminals.

Similarly, to the other circuit to the other side, the posture should be connected to the output of U1 then R6 is of 1.3 mega whereas C7 is 1.22 nano farad. Then C6 is 1 micro and R5 is 150 kilo ohm you say. This is a two-pole bandpass filter. So, if you want to copy, just click on here, so we can see different pop-ups here. One is to just flip our mirror imaging, either one to delete it, other one to rotate and the other one to duplicate, I am just clicking on the duplicate and place wherever it is required to be connected with that is all. So, now we have already made the connections. So, we have to see what is the internal properties of these U1 and U2.

So, to see just click on the Op Amp, click on the Open configuration pane here we can see the properties or the specifications of the generic operation block that we have used it. So since it is a simulation version, and we are not using any you know commercial type of operation amplifiers here the input offset voltage is considered 0 in case if you want to match to the commercial operational amplifier, you can look into the datasheet of the Op Amp and you can provide those parameters at this point.

And input bias current, in this case, is also 0, the input offset current is also 0. And here it has been provided with open-loop gain of 200 kilo-ohms and unity bandwidth of 100 mega input resistance of 10 mega, output 10 ohms. Now, here if you see the Op-amp a positive voltage swing and negative voltage swing it is connected with plus 12 and minus 12. In case if you want to change to plus 15 to minus 15 you can change it to. It depends upon what is the saturation voltage or what is output voltage that you are expecting from the operational amplifier to work or what voltage ranges the op-amp has to be worked with. So, once the changes have been done.

So, then it has been configured to two power supplies plus or minus 15. So, the advantage of going with 3 terminal op-amps in this particular case is that it will automatically provide the required power or required voltages to the operational amplifier. In case if you want to externally provide the power to the operational amplifiers. There is another thing called 5 terminal Op-Amp where other two terminals apart from this input terminal, two input terminals, inverting noninverting as well as output.

It has been provided with another two terminals for providing an external power supply using the power source. So, this is how we can be connected in case if you want to go with a commercially available operation amplifier where the features will remain the same. You can either go with LM324A or 741. If you place LM324A and go to the configuration pain. So, this is a package, this particular specification will be the use of this particular package. So, for a detail understanding of the specifications of this operational amplifier, you can refer to LM324ANG.

So, there are different part numbers depends upon which you would require, you can select that. So, I am deleting this part since our intention was only to go with generate operation amplifier. Now, so, since this is a bad bandpass filter, how do we analyze the working of this filter? So, as we are aware that the design band pass filter will work for a range of 1 hertz signal frequency to 3.7 kilohertz. So, in reality if you want to understand the working of this filter, one good way is by providing an AC signal at the input of noninverting terminal and observing the output voltages.

So, by sweeping the input signal frequency from different frequencies and understanding the output voltage is how much it is being attenuated for the input voltage that gives you the, that gives us what range of frequencies this particular filter will work. Another way to understand this is by providing an by doing an AC analysis sweep which is nothing but it performs a Bode plot analysis where it will provide a different frequency and we can automatically see the attenuation in the output signals. To perform that we can go with a simple AC sweep analysis here.

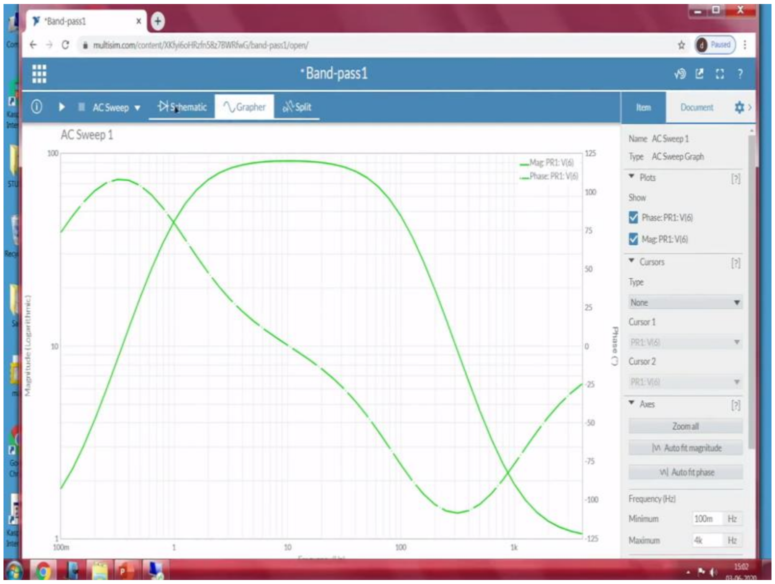
Interactive generally use for normal analysis or a DC analysis. AC sweep where it will automatically do the sweeping of frequencies provided you have to provide from what frequency to what frequency that sweep has to perform. So, since we intended to verify whether this can work for the frequency of 1 hertz to 3.7 K, I will go with the 0.1 hertz frequency to 4 kilohertz frequency. So, points per decade 10 points I am saying particular scale mode then, so, to work with to visualize it, we also have to provide the output terminals to the scope.

In reality, in externally if you want to see the signal, we will be connecting the output of the operational amplifier to the oscilloscope. So, since it is a simulation version, we have to connect to the output terminals. So, we have something called analysis and annotation tap on the left side, which is indicated with the green switch are called as which looks like a probe kind of thing. So, here we have a different probes voltage probe, current probe voltage and current probe voltage reference probe in case if you want to provide some texture. So, for example, first is of bandpass filter we can provide a text.

So, since we intended to analyze the output and providing output at this point and similarly, this is an AC sweep analysis we have to provide because of that we have to go with an AC voltage and other terminals should be connected to the ground. Now, so, even though the set frequency is 1 hitch since we have selected AC sweep as a mode of execution, because of that, it automatically the system or the software will provide the frequencies between the selected frequencies in the previous window.

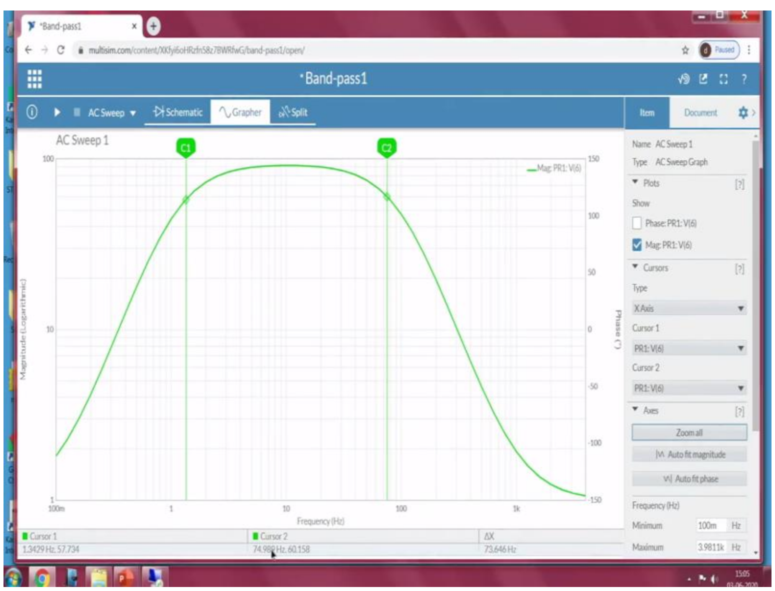
So, somewhere between point 1 hertz to 4 kilohertz and it will automatically plot the output voltage at different you know frequency points and it will be represented in a Bode plot. So, let me run the simulator.

(Refer Slide Time: 24:36)




So, when you go to the grapher, here we can see that the thick green color indicates the magnitude of plot of PR1 V6. So, PR1 V6 is nothing but this 1. So, V6 is nothing but voltage node 6. PR1 is probe 1, the intention of probe 1.

(Refer Slide Time: 25:02)

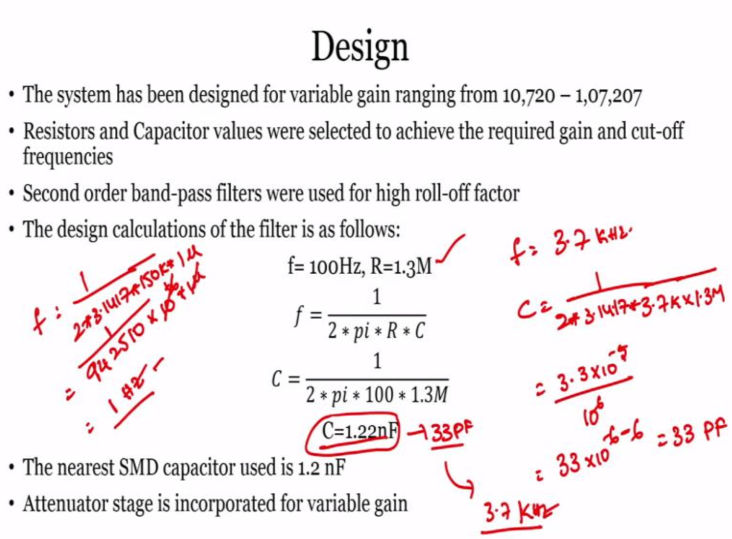



So, this particular plot is an indication of the magnitude at the output of the second operational amplifier, whereas, the dotted line dotted green line is an indication of the phase here we can see. So, further, since our understanding is to find out the cutoff frequencies, upper cutoff and lower cutoff frequencies of the design the filter what I will do, I will disable the phase part-time be I will disable the phase. So, now we can visualize only the magnitude plot. So, the frequencies at which it has ran or it has been simulated is from 100 millihertz that means 0.1 hertz to 3.98 kilohertz.

So, this is the logarithmic scale 1 to 100 where we can see so, if you want to see the cursers, we can create X axis or Y axis curses. So, I am going with X-axis cursor, cursor 2 is also on the PR6. Now, how do we understand the cutoff frequency? So, the cutoff frequency is an indicator of 3 DB line or if I convert into the voltage, voltage form it is root 2 attenuation factor of 1 by root 2.

Now, when I look into the magnitude here, so, 3 DB 1 2 3. So, this particular point will be the 1 2 3 somewhere close to this point is the cutoff frequency of the filter. So, here we can see the cursor 1 is somewhere around 1.38 hedge. So, and whereas, cursor C2, so, 3 DB line 1 2 1 2 3 this particular point 74 hedge.

(Refer Slide Time: 27:48)




So, when we see the calculation design calculations of this particular filter, so, since we have selected R as 1.3 mega and so, the design has been this particular filter has been second-order bandpass filter has been designed with cutoff frequency as upper cutoff frequency as 100 hertz. So, because of that the capacity that we have selected is 1.22 nano farad capacitor. So, we our intention was only for the 100 h frequency in case if you want to go for a little higher cutoff frequency somewhere around 3.7 kilohertz this C can be replaced C and R can be replaced such a way that the R-value and C value when you substitute in 1 by 2 pi R C it will go to 3.7 kilohertz.

So, because of that here we can only see very close to the 100 h value and the gain will be ranging from 10720 to 107207. So, this is achieved by selection of the resistors at this point, R4 and R3 since we have two stages the multiplication factor of both stages. Suppose the same circuit if I want to go for the frequency of 3 point kilo substituting, substituting in the formula, so, we can say so, f is of 3.7 kilo so,

1 /2 *3.14 into 3.7

3.7 kilos them into Are I am considering as 1.3 mega which will be close to $1/2\pi RC$ FC.

So, 1 /2 * 3.1417 *R, I am considering as RC okay R I am considering as 1.3 mega and the C instead of C I am considering F C = 1 / 2 pi F * R. So, which is R I am considering as 1.3 mega

1 = 2 * 3.1417 * 3.7 in 1000 into 1.3 mega. So, which is close to 33 3.3 e power minus 6 or 33 microfarad. So, for a cutoff frequency of 3.7 kilohertz frequency, we have to go with 3.3 into 10 power minus 5 into we have 1 by mega. So, what I mean in the sense.

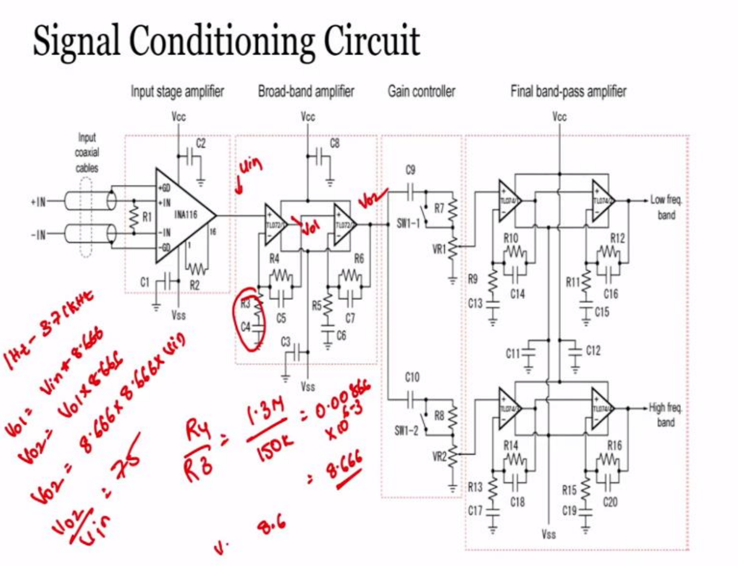
So, since our intention was to go for a frequency of since we required for a frequency of F is equal to 3.7 kilohertz and I will be replacing either a resistor or a capacitor I, I will be choosing 1.3 mega as usual, but instead of that, I will find out what capacitance to be used in order to work for a frequency of 3, 3.7 kilo. So, C = 1 /2 * 3.1417 * 3.7 kilo * R is 1.3 mega. So, when we have calculated 1 / 2 * 3.1417 * 3.7 kilo ** 1.3, we got $3.3 * 10^{-5}$.

Since we have by mega purchase 10 power 6 when they take it up it becomes 33 into 10 power minus 6 minus 6 which is 33 Pico farads. So, replacing a capacitor of 1.22 nano farad with the 33 Pico farads, this particular filter will be designed for 3.7 kilo whereas, for the lower cutoff frequency whereas, on the other, other frequency which is a upper cut off frequency R3 and R4 R3 and C4 is responsible.

So, where we can see F is equal to 1 /2 * 3.1417 * the R in this case we have used of 150 K and C we have used as 1 micro farad. So, when we calculate it 1 /2 * 3.1417 * 150 k * 1. So, we got $942510 * 10^{-6}$

So, the value is  $1\ e^{-6}$. So, this is 1 divided by into 1 micro into micro. So, this and this will cancel this is equal to $1\ e^{-6}$, 1 hertz. So, this is what we got. So, that means 1 cutoff frequency is of 1 hertz other cutoff frequency if we replace the capacitor 1.22 nano farad with 33 picofarad, we will get 3.7 kilohertz. Now, what about the gain?

(Refer Slide Time: 35:19)



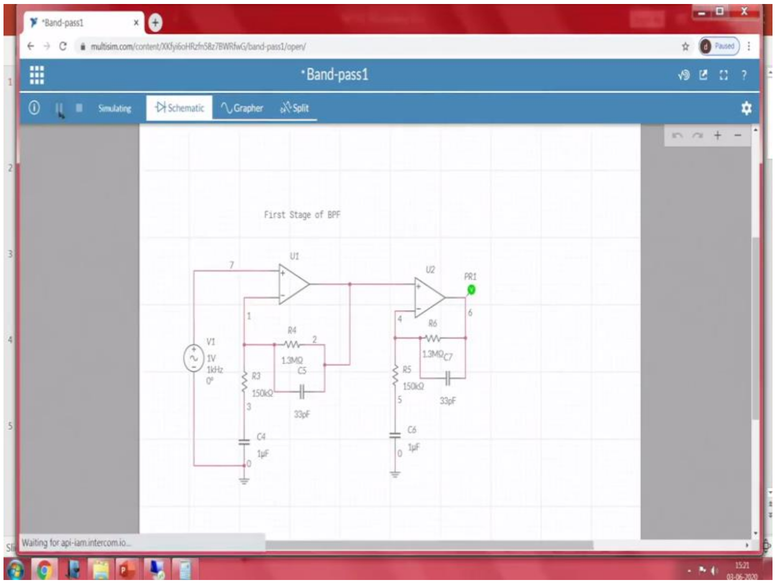

So, when we look into that the gain is nothing but R4 / R3 what is R4, R4 we have selected as 1.3 mega and whereas, R3 we have selected as 150 kilo when they calculate it 1.3 divided by 150 we get 0.008666, 00.00866 into 10 power 6 minus 3 which is nothing but 8.666 gain. Since, we have two one is this case, we have another stage this of replication of the previous stage to increase role of factor. So, advantage of having a higher roller, roll of factor is that higher-order is that it will have a very good roll of factor.

So, you can see the sharp cuts. So, since those two are in connected that means, whatever the output, so, suppose say if this is Vin we have provided with. So, if I say this as Vo1 and this has Vo2 from this we can see that if the input frequency is between 1 hertz to 3.7 kilohertz Vo1 is nothing but Vin into 8.666 whereas, Vo2 is nothing but Vo1 into 8.666. As a result, the total relation between Vo2 and Vo1 will become Vo2 is 8.666 into Vo1, Vo1 is nothing but again another 8.666 into Vin. So, the total gain of this circuit is Vo2 / Vin = 8.666 * 8.666.

So, that is 8.666 * 8.666 so, which is 75. So, now, with that selected R3 R4 and C4 C5 resistances we can achieve a total gain from the first stage as 8.666 and another stage as 8.666.

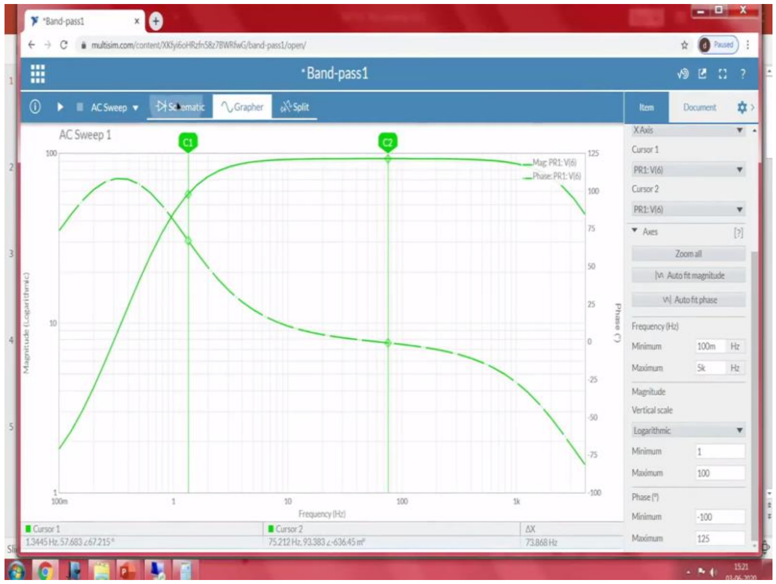
So, total together with a gain of 75. Now, the cutoff frequency if you select a C4 and C5, C5 instead of 1.2 to nano farad if we select it as 33 Pico, we can achieve a 3.7-kilohertz frequency cutoff frequency.

(Refer Slide Time: 38:29)



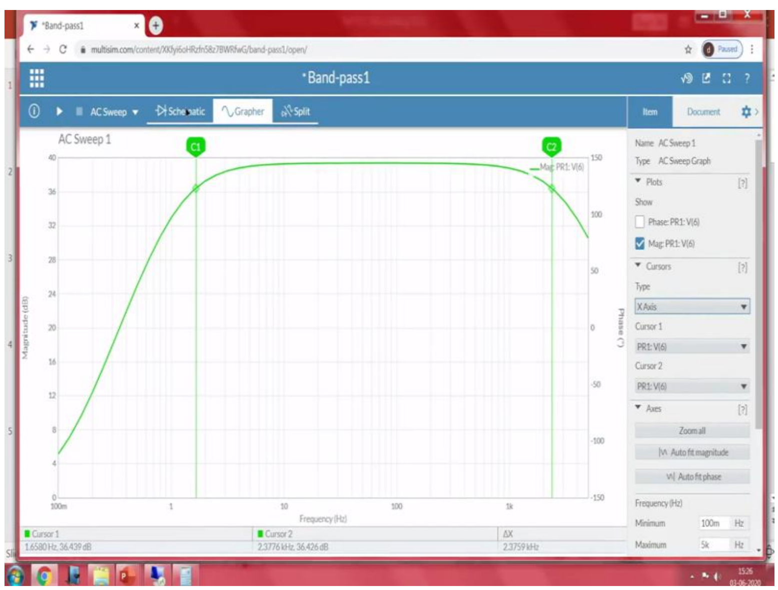

Now, let me replace the circuit, let me change the capacitance value and so 1.22 I will replace with 33 Pico even on the other end 33 Pico and let me run once again it is AC sweep.

(Refer Slide Time: 38:46)




So, I will further change the settings from intro 4k I will go with 5k, I have to change the settings at this point 5k and run.

(Refer Slide Time: 39:11)




2 3 4 5, so I will change the phase, remove the phase, and go with my magnitude. So, I changed the settings as 10k. Now we can see the plot from 100 milli to 10 k hertz. So similar the same way at 3 DB line 1 2, somewhere close to here, similarly even at this point 1 2 3. So, we can see, we can achieve 1.3 hertz and 2.92. So, we can add to the close to what we have targeted for, but

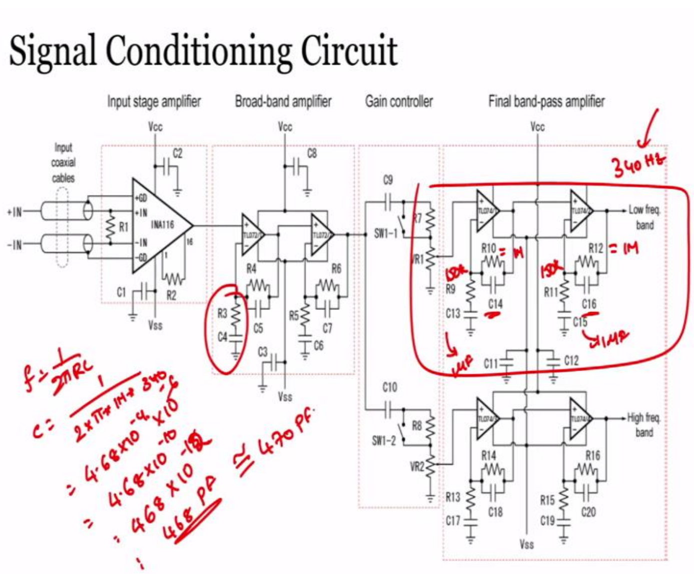
there will always be a manual error because of our calculations and because as we can see that we cannot exactly find out the 3 DB line at this point.

Let me change to Y-axis cursor. So, we can see this is a manual 3 DB. So here I will change it to cursor as X axis so, but 36.494. Go to X-axis 36494 closest value close to this value. So, we got somewhere around 36424 I am adjusting to the value. So, we got somewhere around 1.6 hertz to 2.3 kilo. So, that is what the simulation we have seen.

So however, the idea was to understand whether this particular design can go up to the required frequency. So, what we have seen with this selected by slightly by modifying the capacitance value, we can achieve the required, required cutoff frequencies of the bandpass filter. Now, we will see the second stage of a bandpass filter.

So, now we have seen this broadband, we will go with a final bandpass filter, final bandpass amplifier since we have already simulated a circuit of this one, it is easy for us to implement and verify the cutoff frequencies of this final bandpass filter. So, let us see what are the design consideration based upon the the required frequencies will absorb what will be the cutoff frequencies are based upon the select selected resistance value. We will see the cutoff frequencies of the final bandpass filter in this case.

(Refer Slide Time: 44:20)



So here in this case, I will choose R9 as 150. Similar to what we have chosen in R3 and R5 and R11 also as 150K. To achieve the gain, the required gain of the complete case the R10 has been selected as 1 mega for R9 for assistance of R9. So, R12 is also considered as 1 mega, C13 is considered as 1 micro farad and C15 is also considered as 1 micro farad. So, the selection of 1 micro farad is based upon what is the cutoff frequency that we are looking for. Since this is low frequency band and since we are selector R9 is 150K, since we have already seen at this particular stage, R3 and R4 resistors as which is close to 1 hertz.

So, the use of 150 and 1 microfarad will give the value the theoretical values 1 hertz. Similarly, higher cutoff frequency is of 340, 340 is the target frequency for this particular stage 340 hertz. So, to meet that, we have to find out what capacitance of C14 or C16 to be selected to achieve the cutoff frequency of 340. So, we will go with the same formula

$1 /2 \pi RC$.

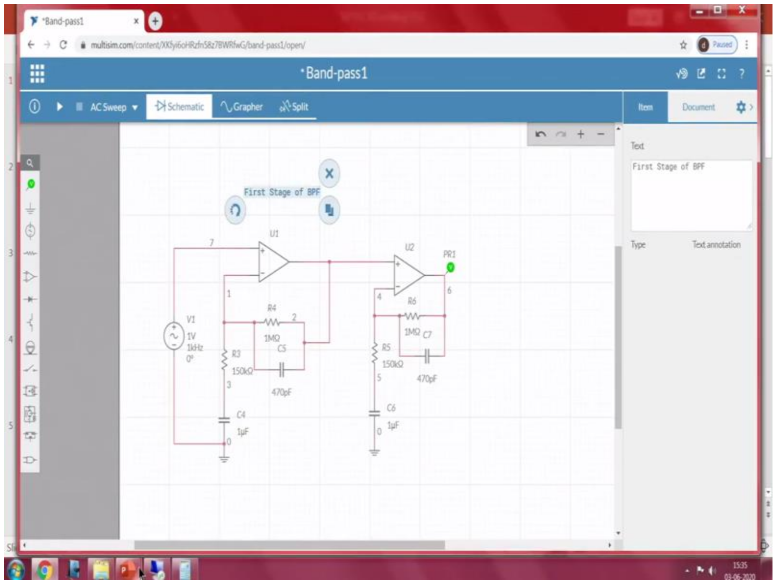
So, since we know what is the cutoff frequency and we have to find out what the capacitance is required, we are substituting the value of R as 1 mega and F as 340 hertz. So, when we calculate it. When we calculated it is nothing but 1 /3.1417 * 1 mega * 340. So, this is

$4.68 * 10^{-4}$.

$4.68 * 10^{-4} *10^{-6}$. So, when we see this, this is $44.68 * 10^{-10}$. So, or I can say 468 so, to so, we got somewhere around $468 * 10^{-12}$. So, which is nothing but yeah 468 picofarad.

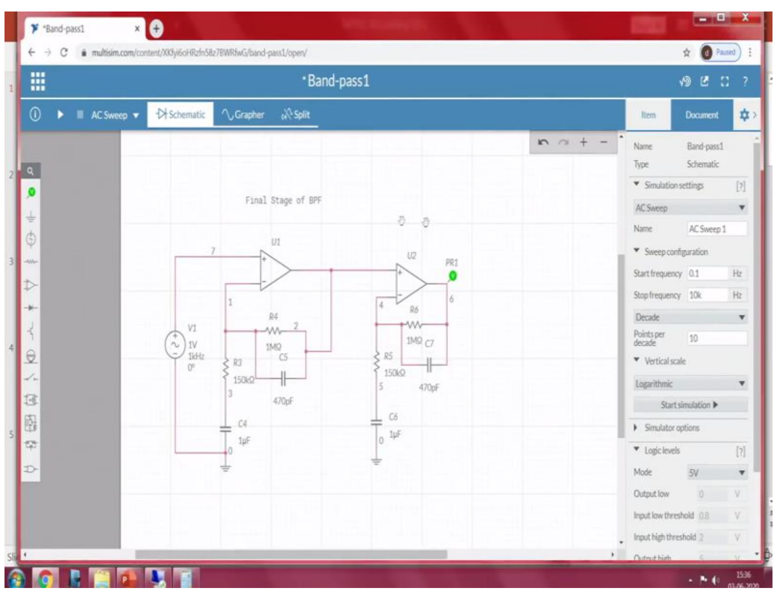
So, in order to construct a bandpass filter with a cutoff frequency higher side cutoff frequency as 340 H. we require a capacitance value for 68 picofarad the close commercially available capacities of somewhere around 470 picofarad. So, I am going to select a 470 picofarad and I will be constructing the same bandpass filter and we will see how exactly the circuit works.

(Refer Slide Time: 47:49)



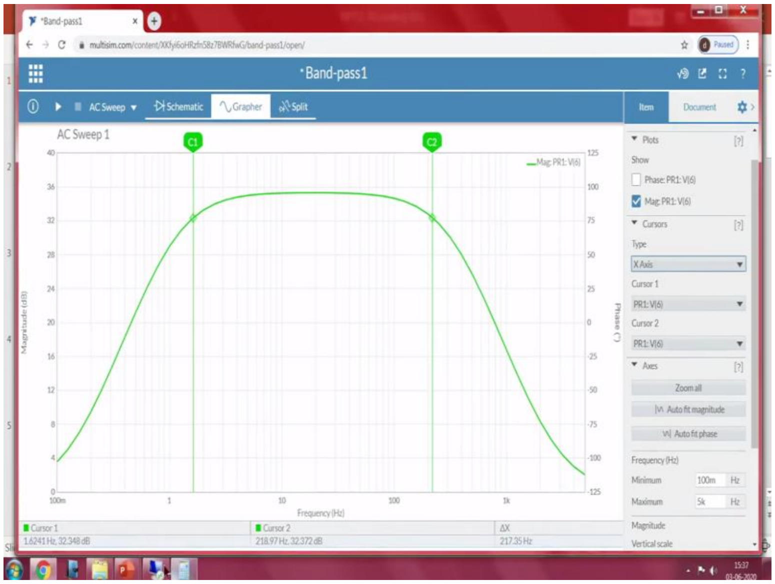

So, I will open online multisim here I am selecting the same bandpass filter which we have seen a last time so simply I will edit R4 R instead of R4 in this case it is of 1 mega I will select 1 mega in this case and C5 which is 470 pico, pico farad, 470 pico similarly even in this case, 1 mega and 470 Pico. Now, when I do the AC sweep analysis, this should be 1 mega 1 mega when I perform AC sweep analysis similar to what we have seen in the first stage of band pass. So, I will rename this particular one as final band pass filter.

(Refer Slide Time: 49:04)




Final stage of band pass filter when we see the AC sweep analysis.

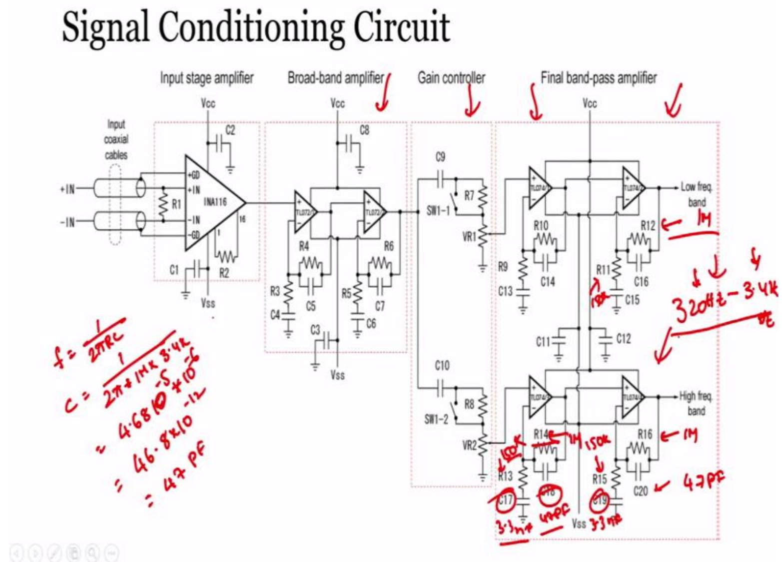
(Refer Slide Time: 49:17)




Similar to the previous one I will remove the phase part and I will go with vertical scale as decibel. So it is easy for us to understand what should be the 3 DB point. I will go with the Y axis. This is a top to the peak value and 3 DB is close to 32.351. So now I will change it to X axis 32.351. So, cursor 1 is almost close to that and cursor 2 will also get to the same. So, now here we can see, we got somewhere around 1.62 hertz to 218, 281 hertz.

So, we have targeted for 340 but whereas we have end up with 218.90 around close to 220 hertz. Similarly, on the other stage.

(Refer Slide Time: 50:42)




So, this particular stage will also find out. So, since in this stage we require 320 hertz to 3.4 kilo. So, in the in this in this high-frequency band stage the targeted frequencies 320 hertz to 3.54 kilohertz. So, since the 3.4 kilohertz is almost equal to the R6 R7 value either we can return to the same R6 R7 value or I will be selecting R6 in this case as 1 mega. So, and R15 as 150 k. So, this will also be 150 and this is 1 mega. So, since, so, we have to find out the capacitance C17 and C18 to match to this particular value.

So, we are designing it now, 1 / 2 pi RC. So, so, C is nothing but

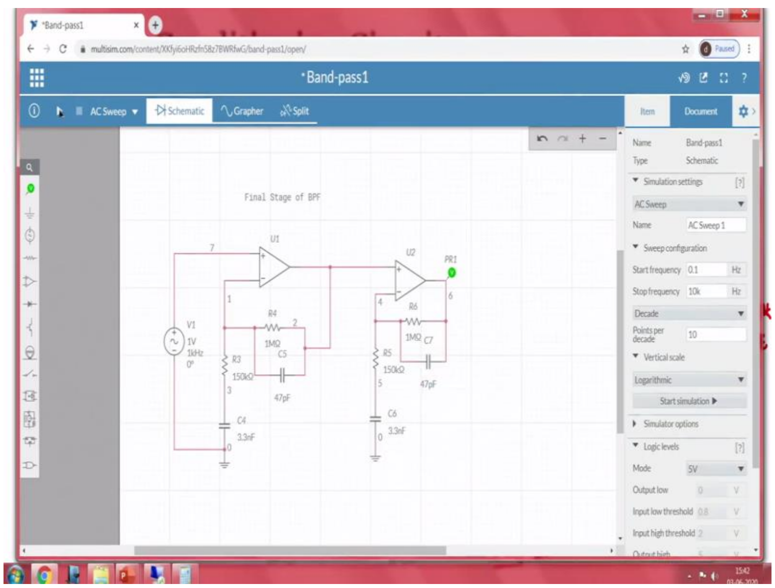
1 /2 pi * 150k. This is 150k k into whereas, frequency lower cutoff frequency a straight 320 hertz. So, when I calculate this so, we are going to calculate this 1 divided by 2 into 3.1417 into 150 k into 320. So, it is 3.3 $e^9$ which is nano farad. So, the closest value is 3.3 nano farad. So, C17 3.3 nano farad and C19 is 3.3 nano farad. So, R16 1 mega if I consider R16 1 mega So, the selection of 1 mega is based upon the required gain that we are looking for the complete stage what is the targeted gain.

So, since individual blocks will also provide the gains. So, the target So, based upon the gains that we have set for broadband amplifier gain controller, the R16 and R12 has been selected in such a way that the total gain, the total gain should be equivalent to the, the targeted gain. So, that is why R12, in this case, is also selected as 1 mega, and R11 is also 150 k. So, that it will not have any mismatch in the gain. Even in this case, the R16 value is also close to the R12 value, so that the gain of this stage is also equal to the gain of this stage, but only the difference is it the cutoff frequencies.

So, now, what value of C20 and C18 to be selected. So, same here, I am substituting R, in this case, is 1 mega and whereas C is 3.4 K so the cutoff frequency. So now, the calculating it 1 divided by 3.1417 into 1 mega into 3.4 kilo, 3.4 kilo. So, the value is 4.68 e power minus 5 4.68 e power minus 5 that is 10 power minus 5 into 10 power minus 6 which is 46.8 into 10 power minus 12 6 minus 5, 11 6 plus 5 11 and 12. So, the closest value is 47 picofarads we can use. So, the C20 is 47 picofarads and the C18 is also 47 picofarad.

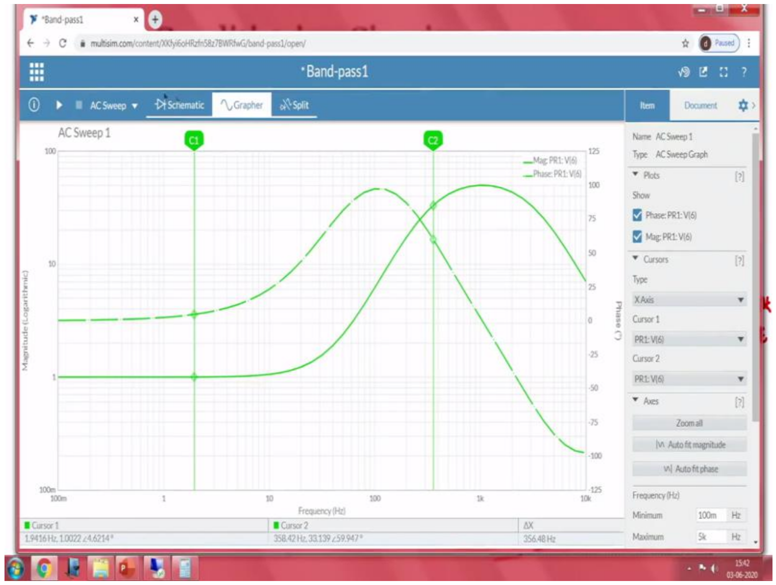
So, if I reconstruct the circuit with the resistance R13 as 150 C17 as 3.3 nano, R14 as 1 mega and R18 as 47 picofarad I can get a bandpass filter with the cutoff frequencies between 320 hertz to 3.4 kilo. So, let us see.

(Refer Slide Time: 55:32)



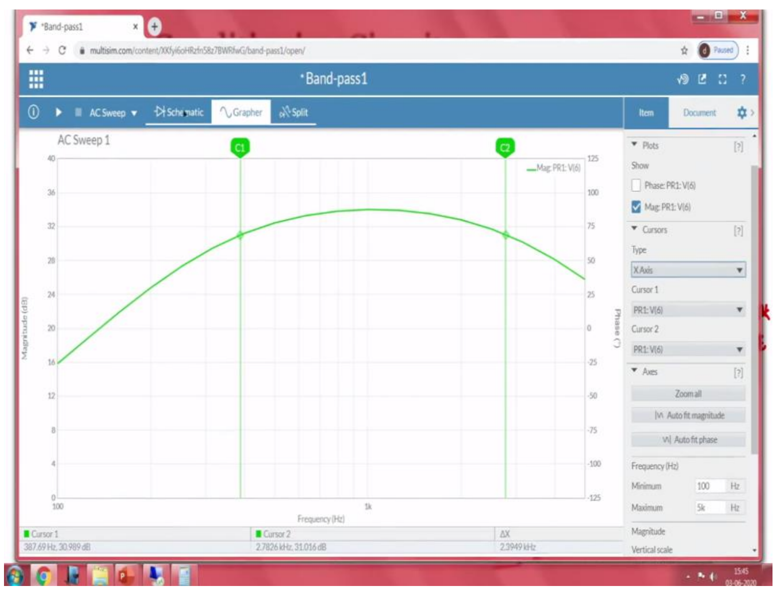

So, now this is 150, this is 1 mega and whereas, this is 47 picofarad and this is the 3.3 nano similarly when in this case 3.3 nano and this is 47. So, now simulating it.

(Refer Slide Time: 55:57)




So, we may have to change the settings. So, since we know what is the targeted frequencies here I will change from 100 hertz or even 100 hertz to close to 10 k,

(Refer Slide Time: 56:21)




Simulating it yes, phase switching off then B then let me put Y axis cuts so that we can analyze DB. So, instead of changing to logarithmic, I am going to decibel point. So, here and minus 3 DB somewhere around close to 31 31 DB. So, now I am changing it to X axis 31 DB. So, here we can see the this particular circuit can pass frequencies between 387 hertz to 2.7 kilohertz whereas target is 320 hertz to 3.4 kilohertz. So, with this, we can understand the implementation of the designing of the circuit using multisim. And we can analyze the circuit the, the cutoff frequencies and the target gain by using this particular circuit simulation software.