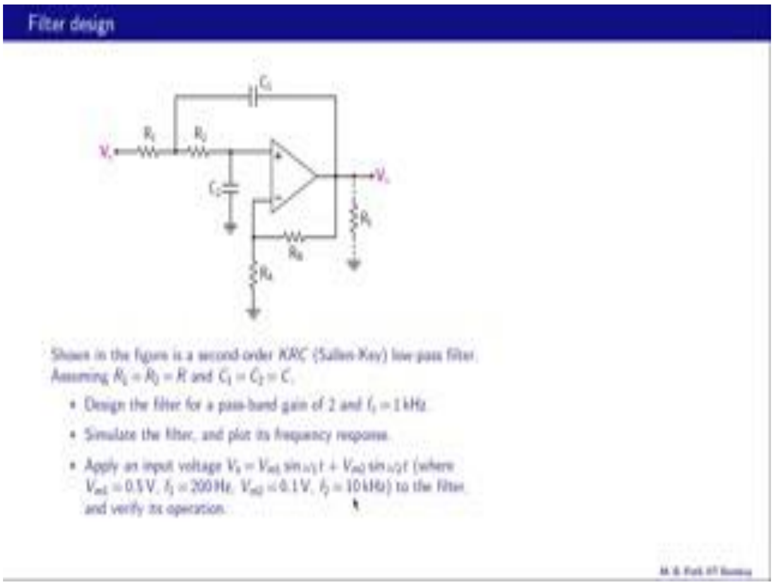
Basic Electronics
Prof. Mahesh Patil
Department of Electrical Engineering
Indian Institute of Technology, Bombay

Lecture - 45
Simulation of op-amp filter

Welcome back to Basic Electronics. In this session, we will simulate an op-amp filter and plot the magnitude of the transfer function versus frequency. It is a good idea for you to repeat the steps shown here on your computer; stop the video whenever required and make sure that you get the same result as that shown on the screen. Let us begin.

(Refer Slide Time: 00:46)

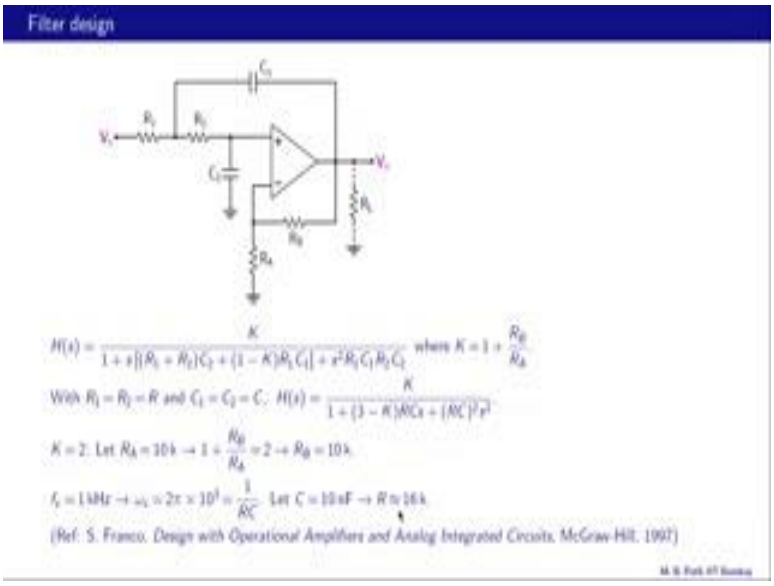


Let us consider this filter problem. Shown in the figure is a second order KRC filter it is also called Sallen-key, and this is a specific filter is low-pass filter it is called KRC because it consists of R C components. And this is the k or gain block. Now, assuming R 1 equal to R 2 equal to R, these two resistances being equal; and C 1 equal to C 2 equal to C, C 1 equal to C 2. Design a filter for a pass band gain of two and a cut-off frequency of 1-kilo hertz. Simulate the filter and plot its frequency response. In particular, we will check that it is a second order filter with the cut-off frequency of 1-kilo hertz.

And finally, apply an input voltage V s equal to V m 1 sin omega 1 t plus V m 2 sin omega 2 t. So, these are two sinusoids. One of them has a low frequency 200 hertz, which is smaller than the cut off frequency; the other sinusoid has a higher frequency 10-

kilo hertz which is higher than the cut-off frequency. So, apply this input voltage to the filter that is when we do circuit simulation and verify its operation.

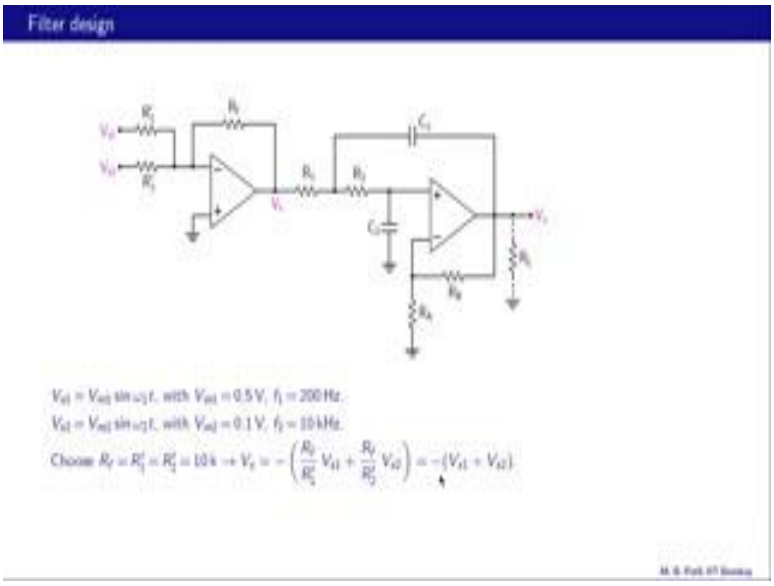
(Refer Slide Time: 02:30)



So, what we should see is that the higher frequency component is filtered out and at the output we get only the low frequency component of V s. And this is our transfer function which we have seen earlier. And in this expression, this k here and here is equal to 1 plus R B over R A and that is the gain of this amplifier section that is a non-inverting amplifier as we can see. Now, let us put R 1 equal to R 2 equal to R, and C 1 equal to C 2 equal to C. And with that H of s reduces to K divided by 1 plus 3 minus K times R C times s plus R C squared s squared. At low frequencies, we can ignore the terms s and s squared because omega is small and therefore, H of s becomes simply equal to K.

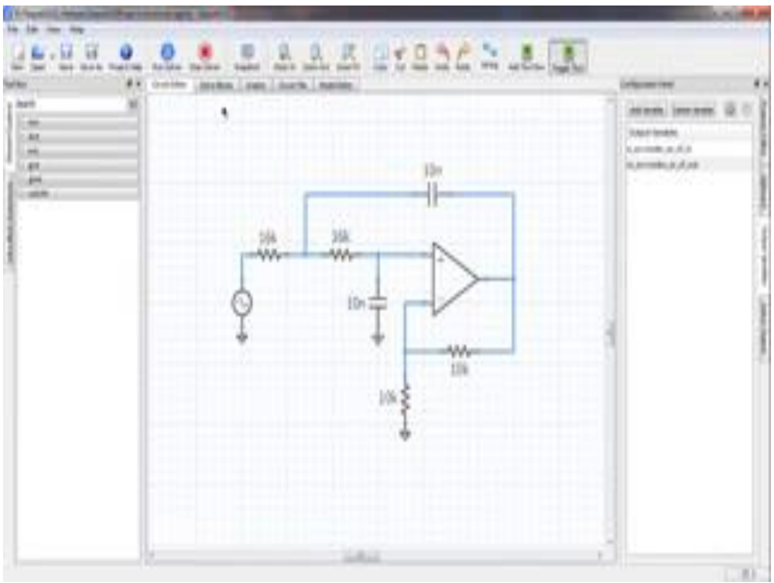
So, when we say we want our pass-band gain equal to 2, what we are really requiring is that K should be equal to 2, so that is our specification. And let R A be 10 k then we have 1 plus R B over R A equal to 2, which means R B and R A must be equal. So, therefore, we get R B equal to 10 k. And note that this 10 k is not very critical; we could have picked 5 k or 20 k for example. Our cut off frequency should be 1 kilo hertz and for this circuit omega c is given by 1 over R c that should be 2 pi times 1 kilo hertz. Let us pick C to be 10 nanofarads and that gives us R of about 16 kilo hertz.

(Refer Slide Time: 04:18)



Let us now come to the second part of the problem in which we have these two sinusoidal voltage sources V s 1 equal to V m 1 sin omega 1 t with the low frequency. And V s 2 equal to V m 2 sin omega 2 t with the high frequency. We want to add these two and then apply to the filter circuit this one. And we can use our op-amp summer to add these two voltages and that is what is shown over here. So, V s is given by minus R f over R 1 prime V s 1 plus R f by R 2 prime time V s 2. And we can choose all of these resistances to be equal say 10 k and then we get V s 1 plus V s 2, this minus sign is not really important in this problem.

(Refer Slide Time: 05:18)



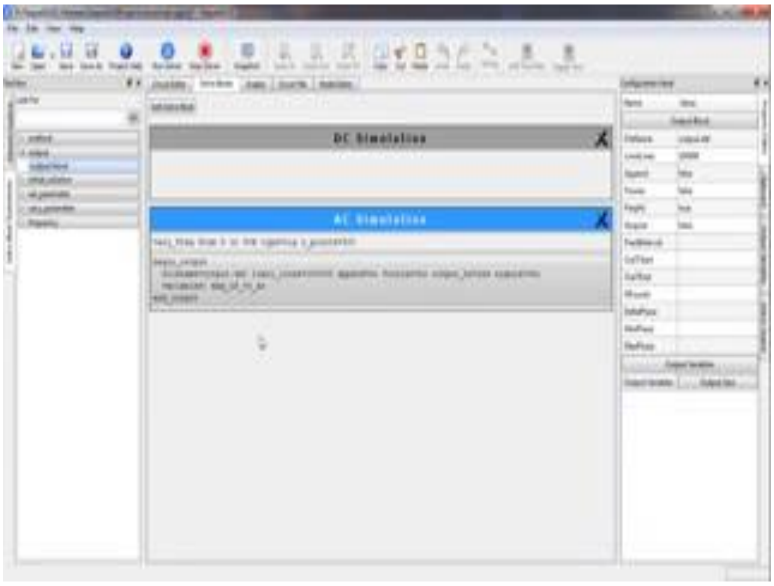
Let us prepare a circuit schematic first. And we begin by getting the components that we require let say we use the 741 op-amp, then the resister, capacitor and ground. We can also take the connector element, which could be useful. So, here is a circuit that we want to the make the filter; and we notice that our op-amp needs to be flipped. So, we select it and then go to flip and then flip vertically. Now, we have plus minus here as we would like. Let us place the connectors, so that it will help us in wiring. We need one here, one here. Let us place the resistances, one there, let us do control C, control V, we need one here one here and this needs to be rotated like that, and we also need one here. We can move some of these things to little bit like that. We need one more capacitor.

So, let us do control C, control V and these needs to be rotated. And we can connect the ground over there like that. We also need a connector here. We can move this a bit. Before we do the wiring, we need to get two more components. So, let us go the panels menu get our toolbox back. We need V s R c A c, we also need ground dummy; we do not need this connector, just remove it. We can now start the wiring because all the components have been placed.

The next step is to assign values to the components this should be 16 k, this is also 16 k, 10 nano, also 10 nano, and this one is 10 k and so is this. We can display this like that. Let us now assign the amplitude for this voltage source let us make that 10 millivolts, and frequency does not really matter, because we are going to vary it anyway. Now, let us name these nodes the input node and the output node that is our in and that is our out. And let us define output variables, the input voltage and the output voltage. And do you want to make these AC quantities that are phasor quantities. So, we right click and then select AC and you notice that it becomes node V ac of in, in is the node name.

Similarly, for the output node we can change these names. Before we prepare a solve blocks, let us look at the help file of this op-amp. And we notice that it has got transistors and some other components and because of that we first need to perform a DC simulation for the circuit, so that an appropriate small signal model is prepared for the op-amp and then follow it up with an AC solve block.

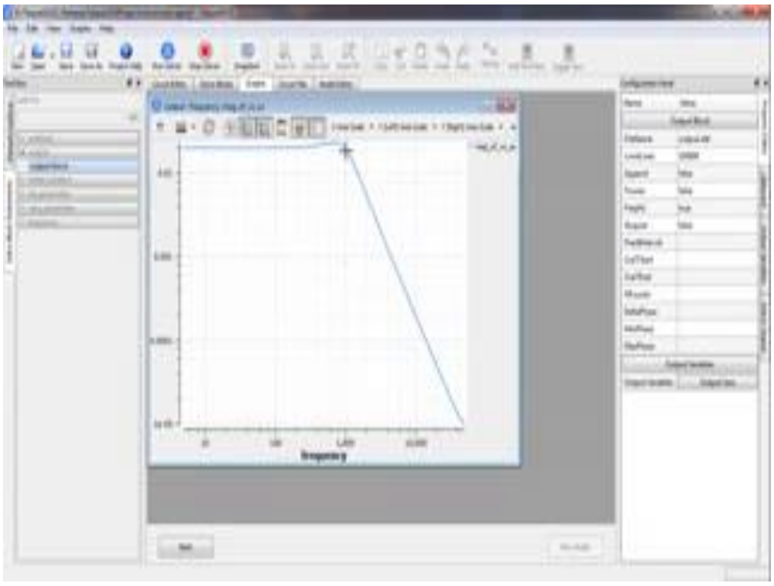
(Refer Slide Time: 10:56)



So, let us do that go to solve blocks, add solve blocks. So, this is the first DC solve block that we would like, this side one more and that should be an AC solve block like that. Next, we need to get the very frequency statement in the AC solve block. So, let us look for that statement, drive it and then click on it. And now you see a bunch of parameters. The very type should be logarithmic because our frequency is going to vary over a wide range number of points should be reasonably large, so that we get a good resolution. The actual frequency values or the range of frequencies is specified in this field here. Let us go from 5 hertz to 50 kilo hertz and that is indicated by 5 comma 50 k. And notice as we type here, this sentence in the ac solve block also gets constructed.

Next, let us get the output block statement. And let us select the output variables now. And we note that because our V i ac and V o ac are phasors the programme has made both the face and the magnitude of these quantities available to us for plotting. In this case, we are only interested in the magnitude of the output voltage. So, we will pick map of V o ac like that.

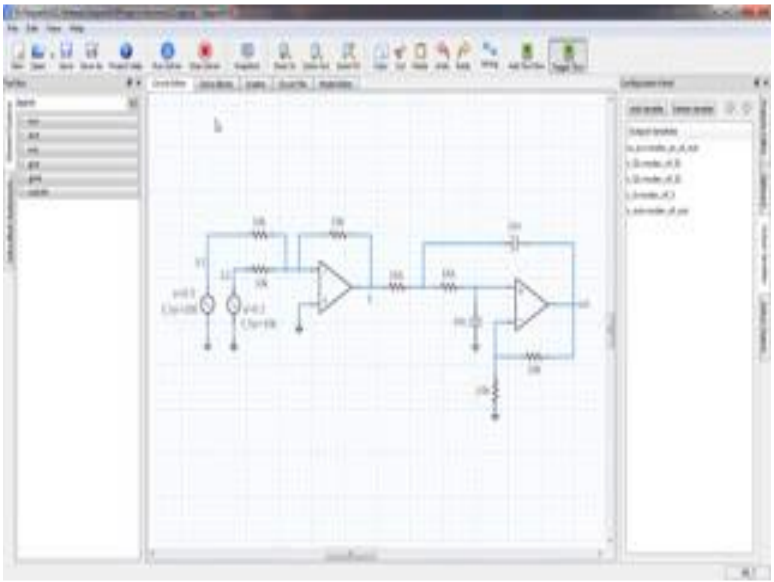
(Refer Slide Time: 12:54)



We can now run the simulation. Let us pick the frequency as the x-axis and the magnitude of the output voltage on the y-axis. And in this case we need to logarithmic axis because of the wide variations both in x and y, so that is what our plots looks like. So, at low frequencies, the magnitude is 0.02, which is what we would expect because our input amplitude is 10 millivolts or 0.01 and the gain is 2, so that is 0.02. Also note that this slope should be 40 dB per decade that is as we go one decade in frequency we should go about two decades in the output magnitude and that is indeed happening. So, this filter does into perform as we would expect. What about the cut-off frequency, we expected to be 1 kilo hertz. So, these two asymptotes should meet at 1 kilo hertz and that also seems to be happening.

Let us come to the second part now in which we want to perform a time domain simulation with an input voltage which is a sum of two sinusoids V m 1 sin omega 1 t V a m 2 sin omega 2 t. The first one has an amplitude of 0.5 volts and frequency 200 hertz the second has an amplitude of 0.1 volts and frequency 10 kilo hertz. So, we need to construct this op-amp summer first, and then apply to input voltages here and then drive the filter this side.

(Refer Slide Time: 14:42)



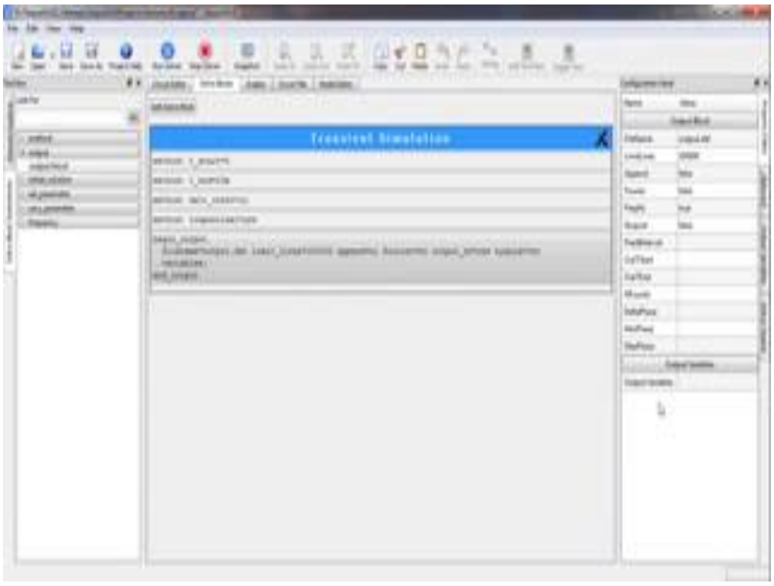
So, let us edit our earlier circuit. The filter we are going to have an op-amp summer here, we can remove the source. And let us first get the components first we require for the op-amp summer. Registers, this is going to be our R f that is a R 1 prime that is R 2 prime. We also need two sinusoidal sources - voltage sources, and we also need dummy grounds like that. Let us now start the wiring, but we will get some connector elements place one there, place one here, place one here, we can now start the wiring.

Let us assign the component values now. All of these resistances are equal 10 k, and all these sources has got an amplitude of 0.5 volts and a frequency of 200 hertz, and the other has an amplitude of 0.1 and frequency 10 kilo hertz like that. Let us display the component values now, right click, add default property text boxes, and let us also display the amplitude and frequency for the sources. So, the way to do that is to right click anywhere on the convex and then select add element property text box and then select the source that we want like that. A is the amplitude, so click on a say ok then a come over here; do that again and select f hertz frequency hertz, and now here both of those. Let us repeat that for the second source like that.

Let us name these nodes you can call this s 1 - source one, you can call this s 2, you can call this s that is the output of the op-amp summer. Let us also display these node names. The way to do that is to right click on an element node, which is connected to that wire, and select add node name text box, and the name appears over there. So, that is our s 1, that is s 2 and that is s like that. Our next step is to define output variables. So, click on output variables and we can click on add variable while pressing down the control key that way we can select more than one output variables. Let us select s 2, s 2, s and the output all right let us rename this like that.

Before going to the solve block section, let us figure out the simulation interval. We have two sources this has a frequency of 200 hertz, and this has a frequency of 10 kilo hertz. So, the larger time period is given by 1 over 200 and that is 5 milli seconds. And suppose we decide to simulate for four of these cycles that means, our simulation interval would be 20 milli seconds.

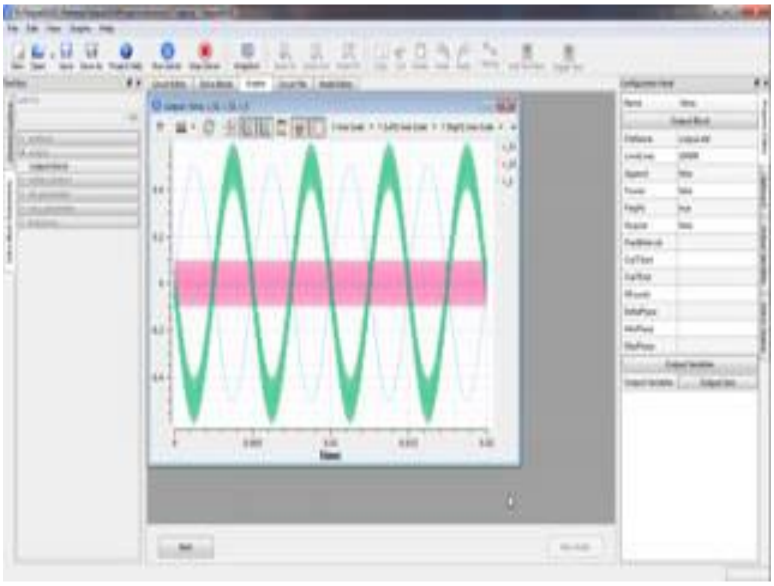
(Refer Slide Time: 20:41)



Let us now go to the solve blocks. We can remove these old ones which came from the filter example. Add a new solve block and we should change that to transient simulation like that. One comment about the numerical method here, we are going to simulate a filter circuit; and in these circuits, the artificial damping that is introduced by the backward Euler method can sometimes cause a problem with accuracy and therefore, what we should really use is the trapezoidal method. So, let us remove that backward Euler option and bring in trapezoidal rule like that. As we discussed earlier our t end could be 20 milli seconds. What about the simulation interval delta t that should be determined by the source with the higher frequency? And that is 10 kilo hertz.

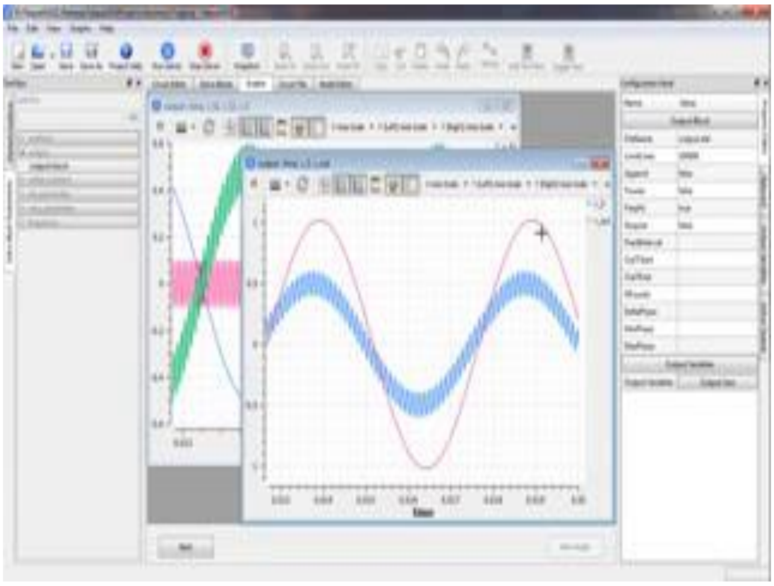
Now, 10 kilo hertz corresponds to a time period of 0.1 milli second or 100 micro seconds and our delta t should be small compared to 100 micro second. So, we can choose for example, 1 micro second. Our next step is to bring in the output block statement and we need to select the output variables. Now, we will choose V s 1, V s 2 V s and V out, this we have defined for the ac simulation earlier for the filter example, and we don t need to look at them now. So, say like that let us now run the simulation. And let us first look at V s 1, V s 2 and V s as a function of time to make sure that our op-amp summer is indeed adding V s 1 and V s 2.

(Refer Slide Time: 23:04)



So, that is what the graph looks like let us expand a little bit. So, here is the V s 1, the low frequency input voltage; this is V s 2 - the higher frequency input voltage, and this our V s - the output of the op-amp summer. So, notice that this higher frequency voltage has got super imposed on the low frequency input voltage giving us V s. And note also that V s has got inverted because our op-amp summer is an inverting summer. Now, let us look at V s and V out together; and from these two, we should able to make out the filtering action of our low-pass filter; it should filter the high frequency component because its frequency is much higher than the cut-off frequency which is 1 kilo hertz and it should retain the low frequency component.

(Refer Slide Time: 24:18)



So, let us plot these two together and that is what we get. Let us expand maybe one or two cycles. So, this is our input to the filter and that is the output of the filter. So, notice that the high frequency component of the input voltage has got filtered out leaving only the low frequency component. And the amplitude of the filter output is higher than the input voltage because our filter has gain of two. Here is a plot along with the circuit diagram the blue graph is V s, and the red one is V out. And as we have commented earlier the filtering action can be very clearly observed from these graphs. Note that there is a small phase difference between the low frequency component of the input and our actual output voltage, but in most applications, this phase difference is really of no particular consequence.

To conclude, we have simulated an op-amp filter from scratch and hopefully you have also got that to work on your computer. In the next class, we will look at a few other op-amp circuits, until then goodbye.