Analog Integrated Circuit Design

Prof. Nagendra Krishnapura
Department of Electrical Engineering
Indian Institute of Technology, Madras

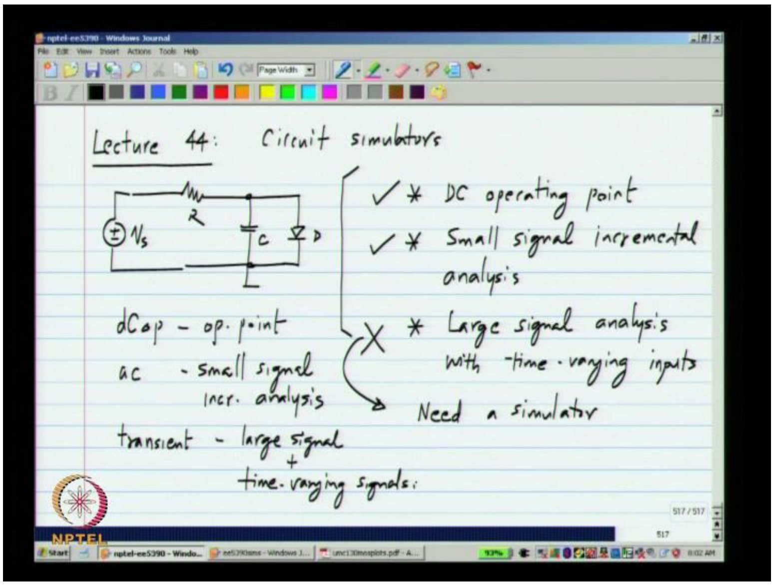
Lecture - 44

Circuit Simulators and Analysis

Hello and welcome to lecture 44 of Analog Integrated Circuit Design, so far we have looked at a number of circuits starting from simple amplifiers, fully differential off amps. Now, we have used some simple models and hand calculations to characterize these, to get an idea of what they do, now this is essential for understanding basics of the circuits then to design them, but finally to characterize the circuits completely. You have to resort to a circuit simulator, because the behavior of each device is complicated, and when you connect a number of devices together, and you put in aspects of all device it is impossible to calculate anything by hand.

So, there are many simulators available, which I will call spice like simulators, spice was one of the simulators that enabled analysis of complicated circuits, and there are a number of simulator available today, which operate on that principle. So, in this lecture what we will do is to look at the kind of analysis that are possible in a simulator, and how to use them to characterize the circuits we have discussed, so far. This is not meant to be a manual on simulators, for that there are detailed manuals for each simulator, which you can consult for the syntax and for the way to use the simulators and so on. This is more to get an idea of principles behind the simulator not in great details, but just enough to use them.

(Refer Slide Time: 01:39)

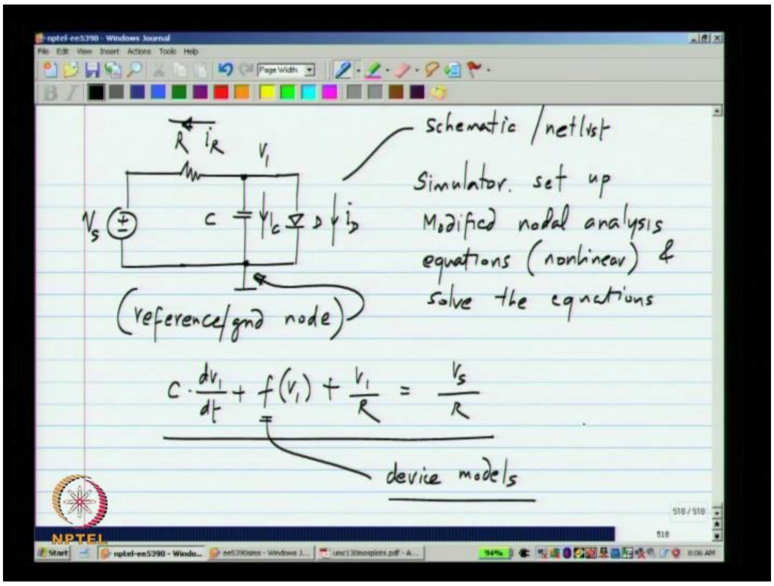


Now, what do we do, when we encounter a circuit, let us say the circuit is this type, I will show some simple circuit for illustration, but this applies to any circuit that we want to analyze, this has r c and a diode and some voltage source. So, what do we do always is compute the DC operating point, now based on the DC operating point, we may do a small signal incremental analysis with an increment applied to various points of the circuit.

And finally, we may also be interested in a large signal analysis with time varying inputs, while doing hand calculations we normally do this in a simple way that is we simplify the models and do this one. And we also do the second one small signal incremental analysis again somewhat simplified and this last thing is usually something that cannot be done by hand analysis except for very trivial circuits. And for this usually any circuit, for which you need to do this you need a simulator, so the basic analysis that are provided by the simulator are these 3.

There is an analysis called d C o p, this gives you an operating point, and there is an analysis called ac, which is for small signal incremental analysis, and finally there is an analysis called tran or transient, which is for large signal and time varying signals. Now, similar to this, basically there is a software to which you have to provide some inputs, and it will do various analysis on the circuit and give you the outputs.

(Refer Slide Time: 04:52)

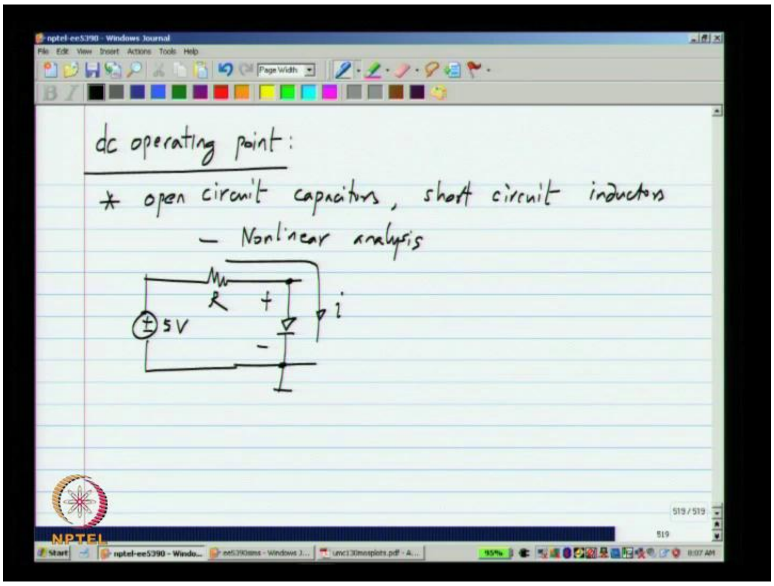


What are the things that you need to give to the simulator, first of all you have to specify in some way what the circuit is, so let us again take a simple circuit, and you have to specify the interconnection of various components, and specify the components. And this is usually done by drawing the schematics with on the software or by specifying the net list, you can consult the simulators manual.

How to do this now what the simulator will do is to setup modified, nodal analysis equations, which are non-linear in general, we have non-linear elements and solve. And solve the equations, now as you know for nodal analysis, your variables are voltages are different nodes with respect to a reference node. So, the circuit also needs to have a ground or a reference node, that is what is the bottom node in this circuit, and finally let us say in this final circuit, where there is only one variable t be solved for v 1. I will quickly write down the expressions the current through the diode plus the current through the capacitor plus the current through the resistor has to equate to 0 by Kirchhoff's current law, and these currents are a function of v 1.

So, what do we have, the current through the capacitor is c d v 1 by d t plus the current through the diode is some non-linear function of the voltage v 1 plus the current through the resistor is v 1 minus v s by r, which is usually written as v 1 by r. And the constants are on the right side, this is the equation for a very simple circuit or a more complicated circuit with, so many nodes you will have a number of different equations and each of it could be non-linear. So, in addition to specifying all these, you also need to specify device models, device models simply say for instance for a diode what phase one wants. So, there are device models available publicly and usually also in many reference books, and you can use these things in order to run simulations on some basic circuits.

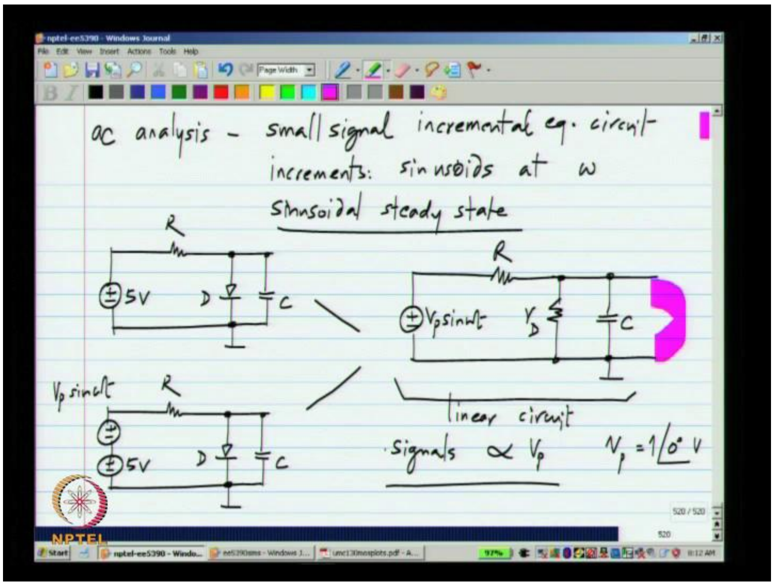
(Refer Slide Time: 07:54)



Let us first look at the dc operating point by definition it is for d c, so what the simulator does is to open circuit all the capacitors, and short circuit the inductors, and basically do non-linear analysis. For our particular circuit the capacitor would be omitted, and we would only have the resistance, and the diode, let us say the voltage got to be 5 volts, then the simulator will calculate the currents flowing in the circuit, and also the voltage at every node in the circuit.

And in this you can get all the information about the circuit, now what do, we do normally while doing a hand calculation for a circuit like this we assume that the diode has point seven volts across it in forward bios. The reason we do that is we cannot solve non-linear equations easily by hand, so we assume that, but if you use a simulator, you get what exactly the voltage is depending on the diode. Depending on the saturation current of the diode, there will be particular voltage across it, and that is what you can find exactly from the simulator.

(Refer Slide Time: 09:41)



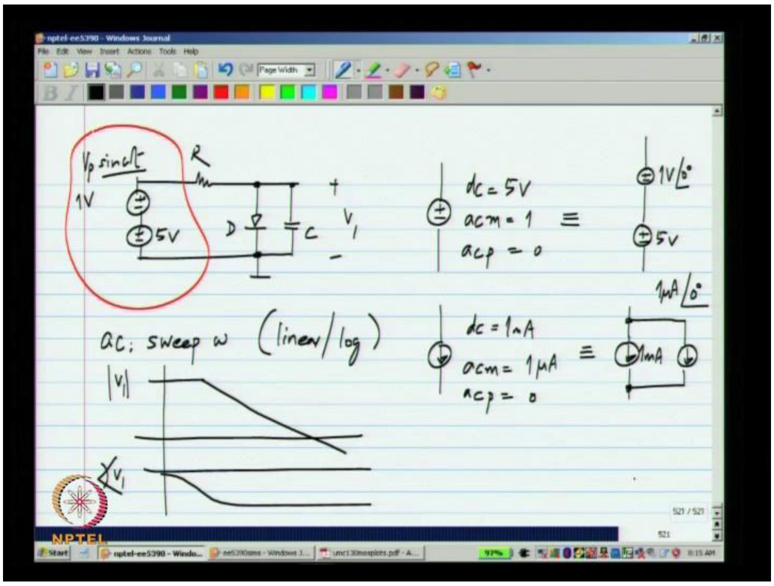
And the next analysis that the simulator provides for is frequently used is known as a c analysis, and this is basically analysis of the small signal incremental equivalent circuit or sinusoidal steady state. Where the increments are not arbitrary, but the increments are sinusoids, add whatever frequency you wish to analyze at, and also it calculates the steady state. So, this basically gives you the sinusoidal steady state, for the incremental equivalent circuit, so what does it mean again let us take a very simple circuit this is what we had and perhaps, now I will have an incremental source in series with this 5 volts. Let me assume that is the case and I have some v p sine omega d and the size of v p the magnitude of v p is such that small signal analysis is valid, in such a case I can use this a c analysis.

So, if I draw the incremental equivalent circuit from these two, I will have an increment of v p sine omega t, the resistor remains as it is, the capacitor remains as it is and the diode will be replaced by 8, so linear small signal equivalent r D. Now, this is by definition a linear circuit so; that means, that the magnitude of v p does not matter, whatever v p you choose the signals in the circuit will be proportional to v p. Now, this is an important point, because one of the other kind of analysis that we do is to apply sinusoids to the circuits, and find out distortion, because we have non-linear elements.

If you apply a sinusoids at a certain frequency at the output, you will also see sinusoids at the harmonics or if you have sinusoids at multiple frequencies, we could see intermodulation components, but this analysis cannot be used for that. This by definition does only the analysis of the incremental equivalent, a linear small signal equivalent circuit and that by definition which is linear, so if you put in a sinusoid at a certain frequency you will only get the sinusoids at the same frequency.

Now, normally one of the things that we determine is the transfer function of a circuit from a certain source to certain nodes, and for that a convenient value of v p to choose is ne volts angle 0 degrees. So, in that case the voltage that you measure here, will give you the transfer function magnitude and phase of the, will give you the magnitude of the transfer function. That is a very convenient value to use, and because analyzing a linear circuit the size of v p does not matter. It is already assumed that v p would be small and you have got the linear equivalent circuit, whether you apply v p of 1 volts or 1000 volts the output will simply change in proportion, now because this is such a frequently used analysis.

(Refer Slide Time: 14:06)



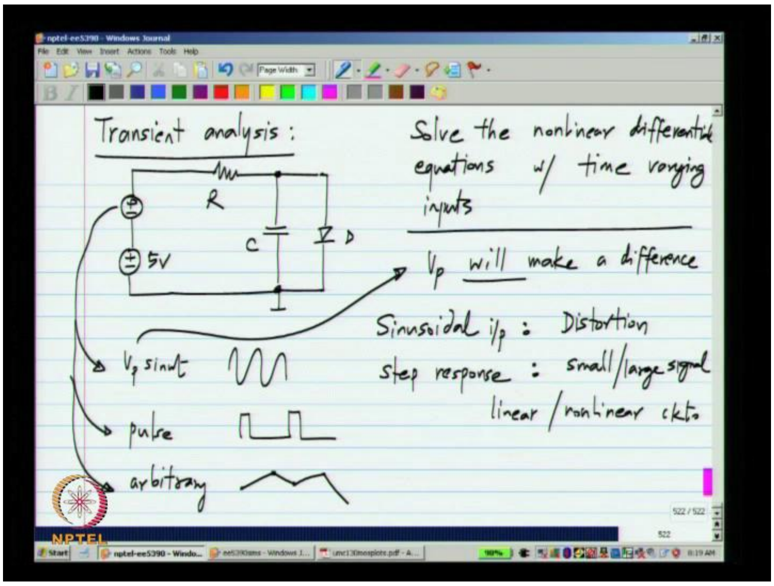
The simulator also provides you certain other convenient features, normally you do not have to specify two voltage sources like this, wherever you have either voltage source by current source. You can specify the dc value, and the incremental values for a c analysis in terms of the magnitude and phase, so let say we want v p to be 1 volt. And I want 1 volt sine omega t, and then the a c magnitude would be specified as 1 and the a c phase as 0, and in most simulators the sine omega t corresponds to 0 phase.

So, this is basically equivalent to having a 5 volt DC source and 1 volt angle DC sinusoidal source or sinusoidal steady state analysis of the small signal incremental circuit. Similarly, you can do the same for the current source, you can specify a DC value of one milliamp, and let us say a c magnitude of 1 micro amp and a c phase of 0 degrees. This is equivalent of having a current source of one milliamp, in parallel with another current source, which has one micro amp angle 0 degrees of sinusoidal a value.

It is a sinusoidal source of 1 microampere 0 degrees also what this a c analysis allows you to do, is to sweep the value of omega, because here we have not specified the value of omega. Once, you have the incremental equivalent circuit like this, you can calculate it as a function of omega, this is what usually you are interested in plotting broader plots, and in the magnitude in phase response.

So, you can sweep omega in different ways linear log and so on, in a number of ways and plot the results, so what you get out of this analysis is the magnitude of the voltage. So, let us say across the capacitor, you have v 1 you can plot the magnitude of 1 and the phase of v 1, so it could look like that and then phase could look like that a very useful analysis. You have already seen how often we do small signal incremental analysis of circuits, and that is done using a c analysis from the simulator.

(Refer Slide Time: 17:03)



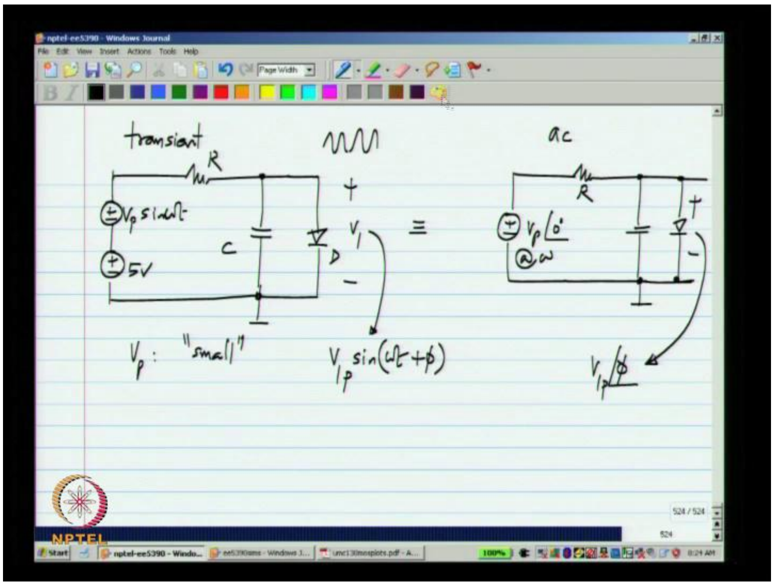
And finally, we have transient analysis, which as I said earlier is the real reason to use the simulator, because this is almost impossible to do by hand for anything,, but the most trivial circuits. Let us say again I have a particular case, again I am looking at here, I have this 5 volt, and I could have some increment, but this is not necessarily a small increment, this could be anything. I am just showing it like that, the increment could be anywhere else in the circuit, and by the way this increment also could be let us say a sinusoid of some frequency or it could be something else.

It could be a pulse of some shape, and also these days similar tests will allow you to specify arbitrary wave forms using some piecewise linear definition, you could specify anything you want exponential, sinusoids, pulses and, so on. All the commonly used shapes will be there, and you can also specify arbitrary shapes, so what the transient analysis will do is solve the non-linear differential linear equations with time varying inputs. And in this case, because you are solving the non-linear differential equations, for instance if you put a sinusoid the amplitude of the sinusoid will make a difference.

And why do we use this transient analysis with the sinusoidal input, normally you would try to calculate the distortion produced by the circuit, that is you are putting a sinusoid at a certain frequency. And you do the analysis, and you can by taking the Fourier transform of the output, whatever the relevant output is you can determine the strength of the harmonics.

As, you know you have the amplifier for instance the harmonic distortion is a very important criteria and that is computed by doing a transient analysis, and another thing of interest usually is step response and this is computed by using a pulse of appropriate size. And you could do this for small or large signal and for linear or non-linear circuits, in case of linear circuits step response characterizes the transfer function, in case of non-linear function also you may be interested in, how the circuit behaves when you apply a step. And of course, you could apply any kind of signal, and also evaluate the response, for instance one of the interesting thing is.

(Refer Slide Time: 20:48)



Let us say you have the circuit, that is powered by some power supply, the second is a commonly used analysis, one of the interesting thing is how the circuit starts up, when you power the circuit it does everything come up correctly or not. So, for that what is normally done is the supply voltage is stepped up, ramped up to certain rate from 0 to it is value, whatever it is let us say 3.3 volts.

And then you examine what happens on the circuit to see that everything comes up, now again this is a non-trivial thing, because of non-linearity and complex feedback loops. Sometimes, there could be a situation, if you analyze the circuit with the 3.3 volt supply everything looks fine, but if you start it up from 0 and go all the way to 3.3, it does not reach the right state, there could be some complicated feedback loops which latch up to some other value and so on.

So, this is again something that is very important this mimics reality, so you have a circuit, you put a battery on it and then you turn on the circuit, you would like to see if it turns on the correct state or not, and how are all these things related. So, let us say, we have the d c operating point that we analyzed for the circuit, let us say for 5 volts, let us say you make a small change to one of the sources let us say 5.001 volts. Now, the result the difference that you get between when you apply 5.001 volts, and 5 volts is the increment, and that will correspond to what you get from the a c analysis for an increment of 0.001 volts at omega equal to 0.

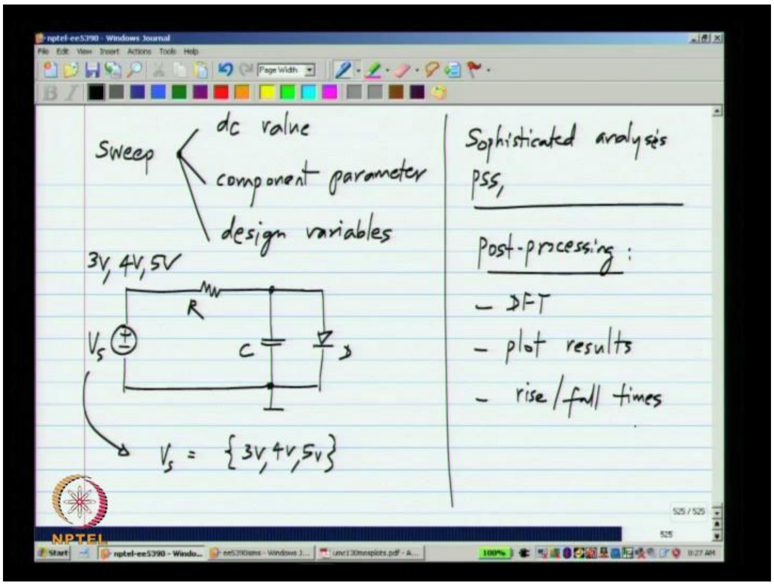
Here, we are talking about omega equal to 0, so if you do the DC operating point analysis for let us say a small voltage source or current source, the same result can be got from a c analysis at 0 frequency. And this will be valid as long as the increment in the DC values are rather small, similarly let us say you have a transient analysis setup, and let say v p is small, what does it mean for it to be small.

It means that the small signal assumptions are valid, in such a case let us say you determine the voltage across v 1, and it will also have a sinusoidal component, which will have a different amplitude from v p. And it will also have a space shift compared to the input sinusoidal, now what you get from this analysis would be the same as what you would get out of a c analysis, for an increment of v p at omega. You apply v p angle 0 degrees set of frequency of omega, in case of a c analysis you have replaced this nodule, and similar way you would have replaced this by it small signal link incremental resistance, and you do a c analysis on this circuit.

Let me put on the original circuit, and find the magnitude and phase of this, you will get some amplitude and some phase, and here what you would have got would be v 1, let me call this v 1 p sine omega t plus psi. So, transient analysis with a sinusoidal input, and a very small amplitude will give you the same result as a c analysis, now a c analysis by definition is linear.

So, that will correspond to what happens to the very small input signal, now these are all things that you can try out especially, when you are beginning to use a simulator just, so that you get used to what different analysis do and also how to do the analysis themselves. Now, in addition to these the simulator provides many convenient facilities, such as sweeping either the DC value or some component parameter or you can have design variables in the circuit. You can consult simulator’s manual to see how to do these things, but the point is you can run the simulator only once, and let us say you want to calculate the DC operator point of the same circuit that we had all along.

(Refer Slide Time: 26:05)



Let us say for different values 3 volts, 4 volts and 5 volts, you do not have to set up the circuit four times, and then run the simulator 3 times, you can specify that this voltage takes on 3 values. The simulator will go on and analyze it 3 times, the 3 different values, and give you the result, and these are all convenient features this is just, so that you have to do less work and you can get the result more conveniently.

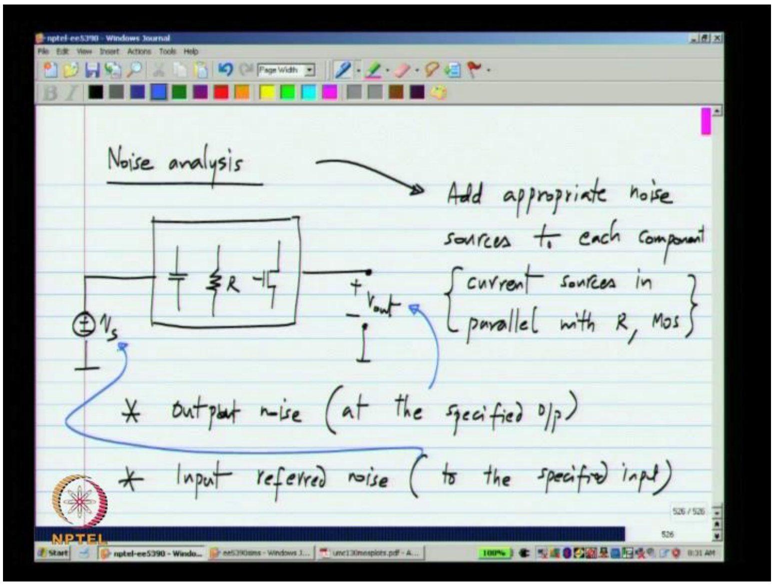
Similarly, you can also change the component parameters and see how the circuit behaves, sometimes you may also want to vary some component value in order to optimize it, or in order to just study the behavior of the circuit. So, that also is possible, and in addition to these things there is a number of more sophisticated analysis, such as periodic steady state and a number of others, which are suitable for specific circuits. There are a number of circuits that operate periodic inputs, and attain a periodic steady state for that periodic steady state analysis is useful.

So, again these do not fall under the basic analysis and we will not rate them here, but you can look at simulator’s manual and the extensive information, that comes with it to find out about details. And besides analysis what you can also do in most cases is convenient post processing, this could include for instance for computation of distortion you have the Fourier transform, the discreet Fourier transform that is available.

And you can plot the results in number of ways number of convenient ways for visualization, and then for instance you can compute the rise fall times of step response and so on. So, these are all convenient additional features to the simulator, now the simulators have evolved over the last forty years or, so maybe even more, and by now they are extremely sophisticated.

So, you can analyze very, very complicated circuits with thousands of transistors or even more using the simulators, for instance the tablet p c on which I am giving these lectures has a number of integrated circuits, which are also simulated using the simulators. But, first you should get familiar with the basics of the simulators, for that you can try the DC operating point a c and transient analysis on simple circuits. And make sure that they conform to what you calculate by hand, after that you will get some confidence in using the simulators you can go on to analyze more complicated circuits.

(Refer Slide Time: 29:45)

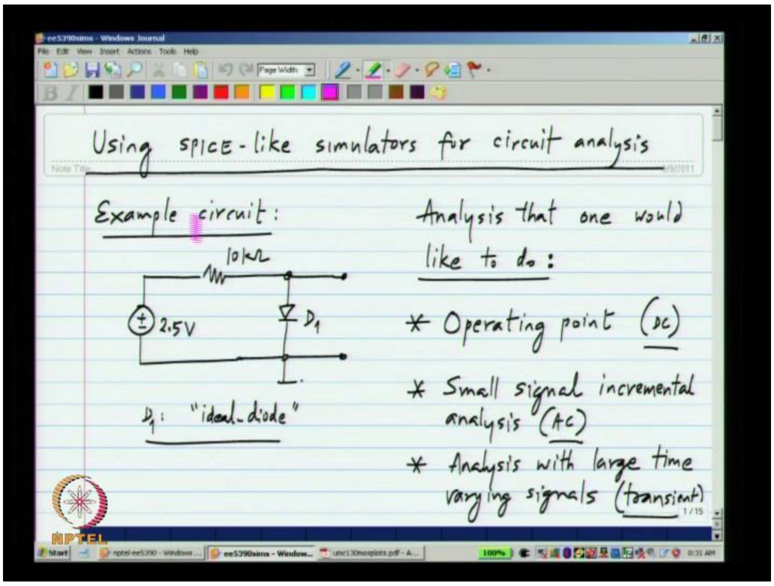


So, that is about the basics of the simulators, and one other type of analysis that is very commonly used, and very convenient is noise analysis, you have a circuit where they have a number of components just like capacitors resistors mos-transistors and so on. What noise analysis does is to add appropriate noise sources to each component, for instance this could be current sources, and parallel with resistors and mos transistors.

And you have to specify an input port or a voltage source, and a pair of output nodes, and you need this, because this analysis will calculate the output noise add the specified output, and also the input referred noise refer to the specified input. So, you may have a number of inputs and outputs in the circuit for each output, you have to do the analysis separately. You should specify the output at which you want to compute the noise you should also specify the input source, referred to which you want to calculate the input referred noise.

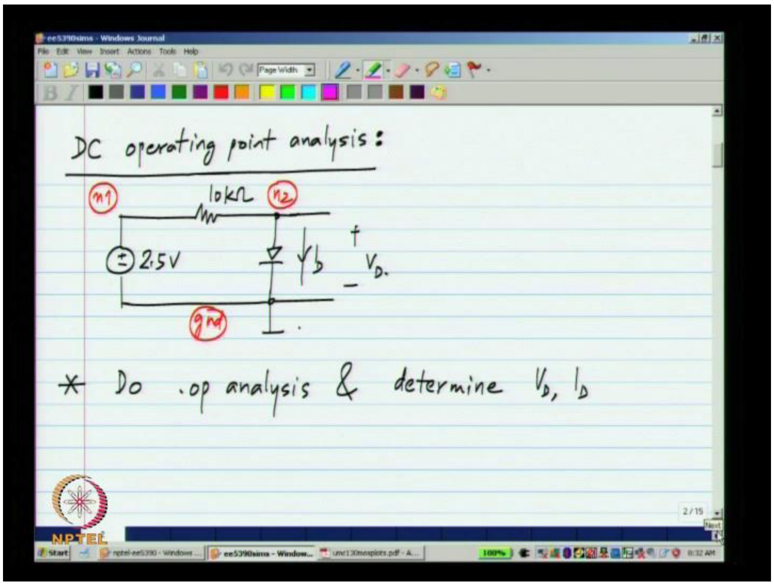
If you recall the input referred noise is an equivalent noise, that if you apply the input of a noiseless equivalent circuit, you will get the same output noise. And also one other feature of the more simulators is that, you cannot specify a current as the output quantity what you can do is use an ideal current control voltage source. And specify that voltage as the quantity, and that will give you the noise in the current, so again by practicing with a few circuits you will be able to get a hang of this one.

(Refer Slide Time: 32:20)



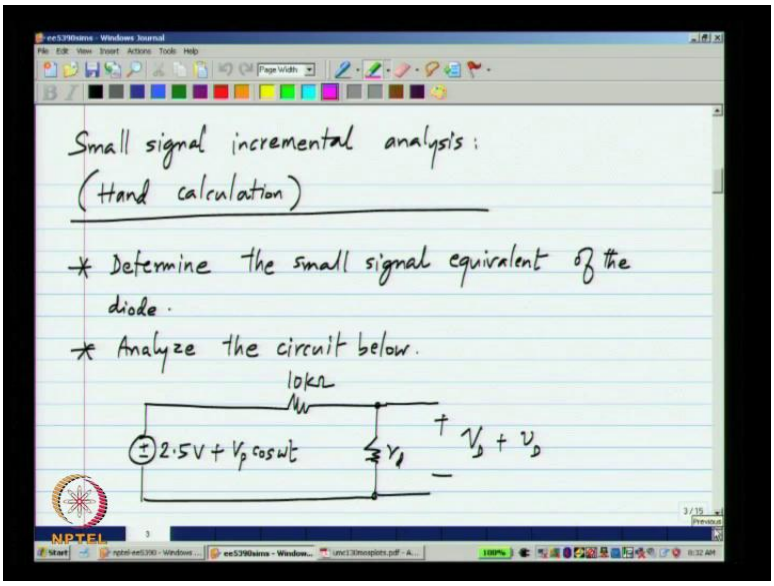
So, just as a quick recap here I have shown an example circuit, so we want to do operating point analysis, small signal a c analysis with large time varying signals.

(Refer Slide Time: 32:38)



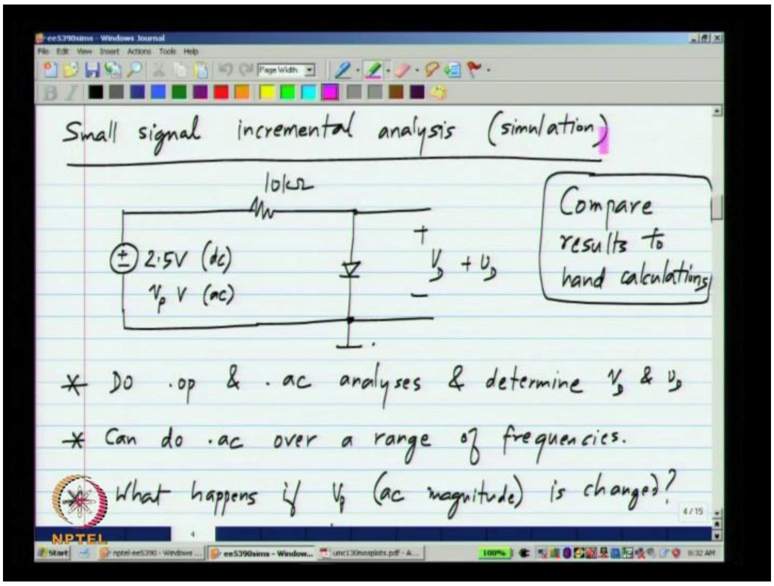
So, what I suggest you do is take it as an exercise, and first of all you can do a hand analysis for assuming points on volts do not drop across diode. And then you do a operating point analysis DC o p analysis, and find the diode voltage and current and see how they differ, because the voltage cross the diode will not be exactly be point seven it will be somewhat different.

(Refer Slide Time: 33:03)



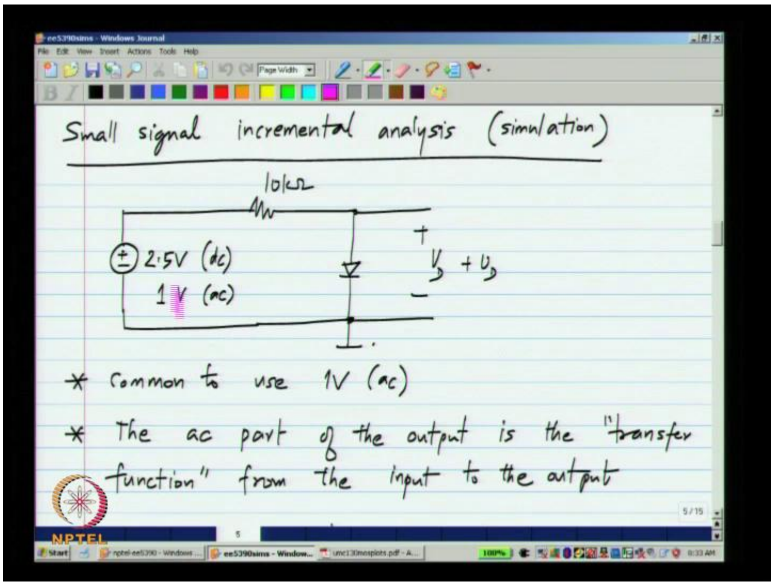
So, again you can do the a c analysis, let us say first you do it by hand, you calculate what you get out of the linear small signal equivalent circuit, where you replace the diode while it is a small signal equivalent circuit, and find out what comes out the incremental v d that comes out.

(Refer Slide Time: 33:26)



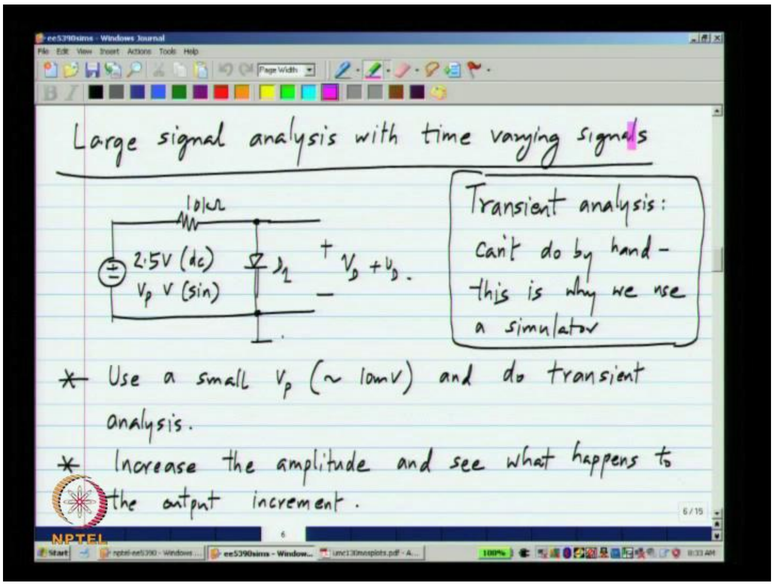
And then you do the simulation, you do the DC operating point and the a c analysis, and compare the results, you can also do an a c analysis for a range of frequencies. And you can also try and see what happens, if the amplitude that you specify for the a c analysis is changed. As, I mentioned earlier his is for a linear circuit, that it does the a c analysis, so that the output should simply be proportional to the input and of course, you can also compare the result to the hand calculations.

(Refer Slide Time: 34:05)



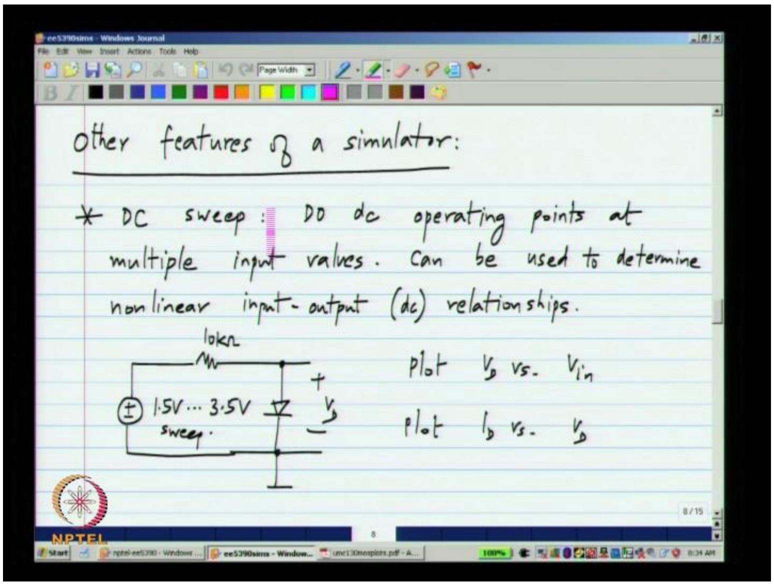
And like I said before the amplitude here does not matter, so it is very common to use one volt of a c, and what you get out is the transfer function of the circuit from this source to this output.

Refer Slide Time: 34:24)



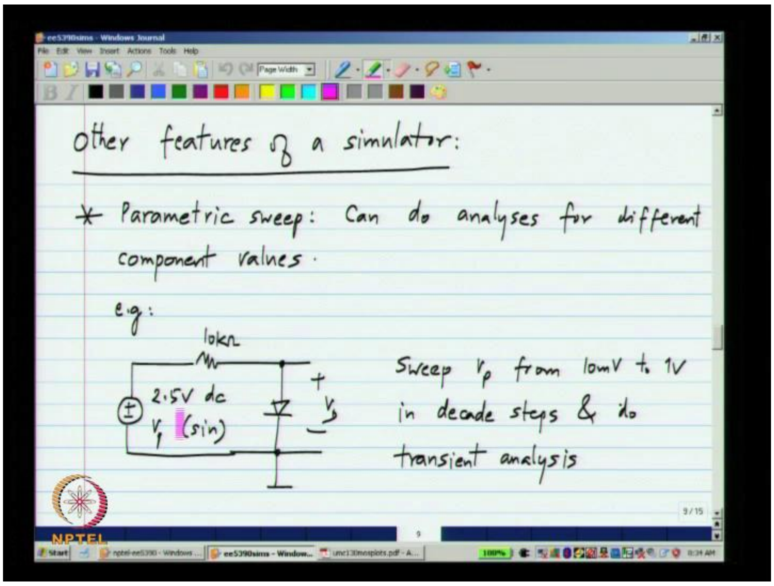
Again the next thing is to try and do a transient analysis with a simulator, so you first apply a small sinusoidal source in addition to the DC value, and do the transient analysis. And you go on increasing the amplitude v p and see what happens to the output wave form, in this case the result will be different from a c analysis and that is interesting in itself. And what you can also do is to compare the results when the results are very small, to what you calculated by hand and also what you calculated by using a c analysis.

(Refer Slide Time: 35:07)



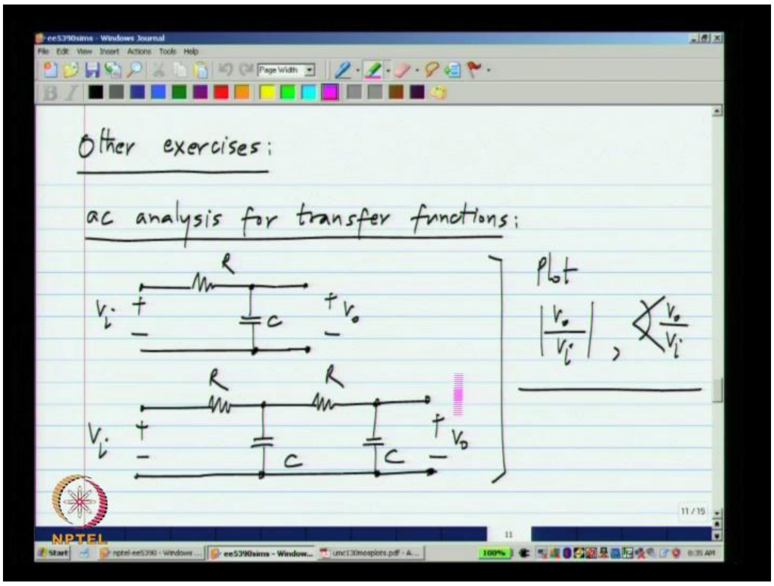
Like I mentioned earlier you can do a DC sweep, for instance you sweep the values of these voltages and plot the diode voltage across the input voltage, you can also plot the diode current across the v d, and get the characteristics of the diode.

(Refer Slide Time: 35:22)



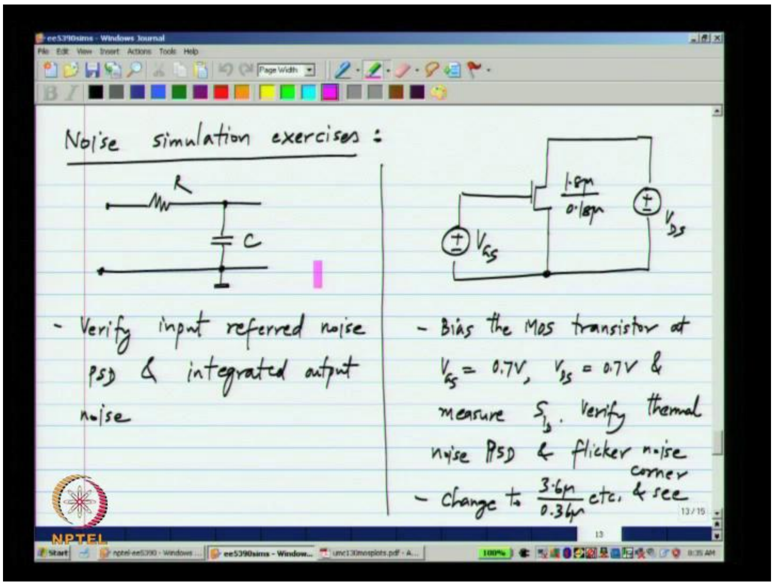
Similarly, here you can do a transient analysis or an a c analysis by sweeping the value of d c, so for different values of DC you can do the transient or DC analysis.

(Refer Slide Time: 35:37)



And you can take a linear circuit like this and calculate the transfer function, that is you plot the magnitude of v naught by v i and angle of v naught by v i, which together give you the sinusoidal steady state response, or basically give you the transfer function the circuits or sinusoidal inputs, this is something that very commonly required.

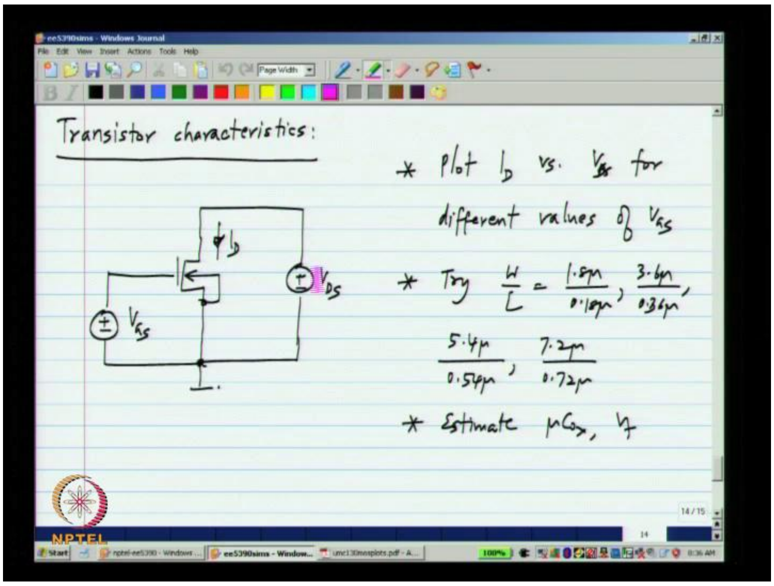
(Refer Slide Time: 35:56)



You can do the noise analysis for a simple circuit like this, we already know by hand calculation what the result must be for this particular circuit, and we know what the integrated noise across the capacitors, you can also verify this using simulation.

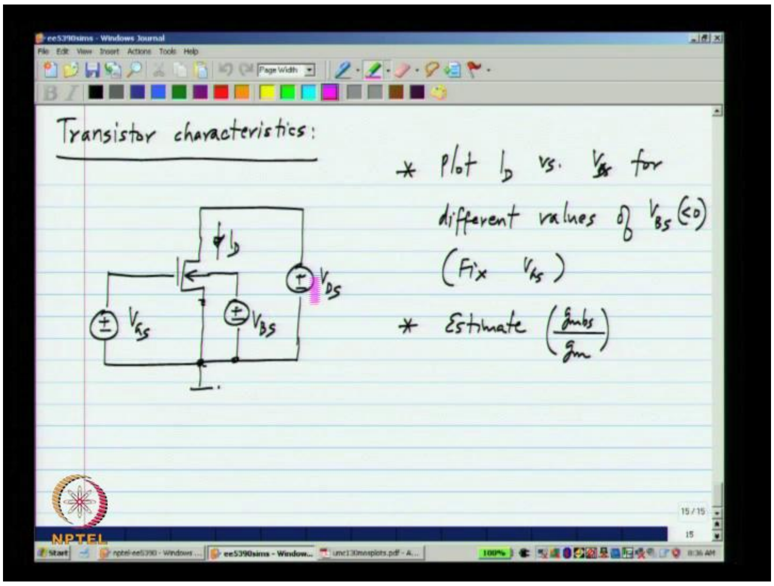
Similarly, for a mos transverse circuit you your bios has a certain value, that has v g s point 7 volts, and measure the special density of the current noise of the mosfet. And you can also plot it, and find out the flicker noise corner and the thermal noise become equal to flicker noise and so on.

(Refer Slide Time: 36:43)



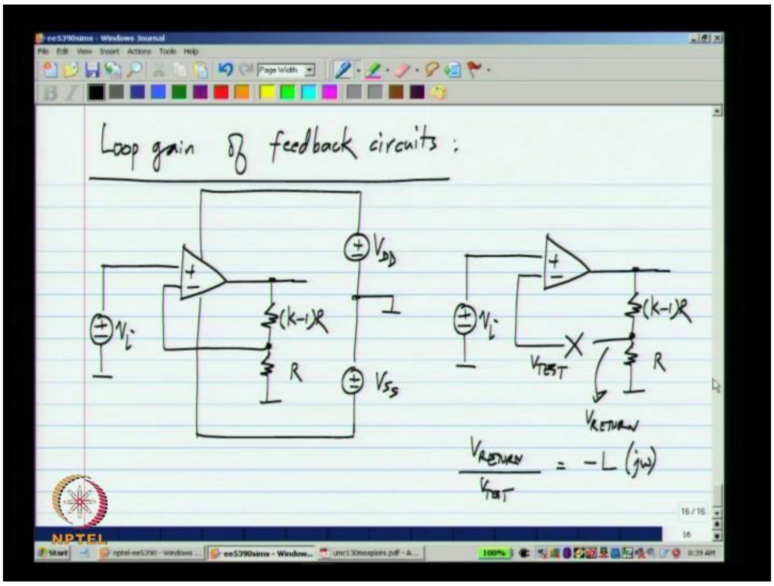
You can also change the size of the transistor, and see how the noise behaves, and you can plot the transistor characteristics, to get approximate values of let us say mu one c mos the threshold and so on. So, you can plot i d versus v d s for different values of v g s, also plot i d versus v g s, you have to chose, which one to sweep whether v g s or v d s and you try it for different w by l and so on. So, again first of all this will help you to get used to the simulator, and also for the transistors you have it will let you estimate the characteristics.

(Refer Slide Time: 37:09)



And in addition to v g s and v d s you know that the bulk source voltage also has an effect, so for a fixed v g s you can sweep v b s and see, earlier I said that the bulk adds as another gate to the mos transistor. You can verify using this exercise, whether it is indeed the case, now please go through all the exercises that I just mentioned. As, I said it will first of all let you get used to the simulator, because it takes some time to get used to it then get the simulator to behave the way, you want and it will also help you confirm what you already know about circuits using simulation. Now, that we have the simulator one of the important things that we need to calculate besides transverse function and so on is loop gain of feedback circuits.

(Refer Slide Time: 37:52)

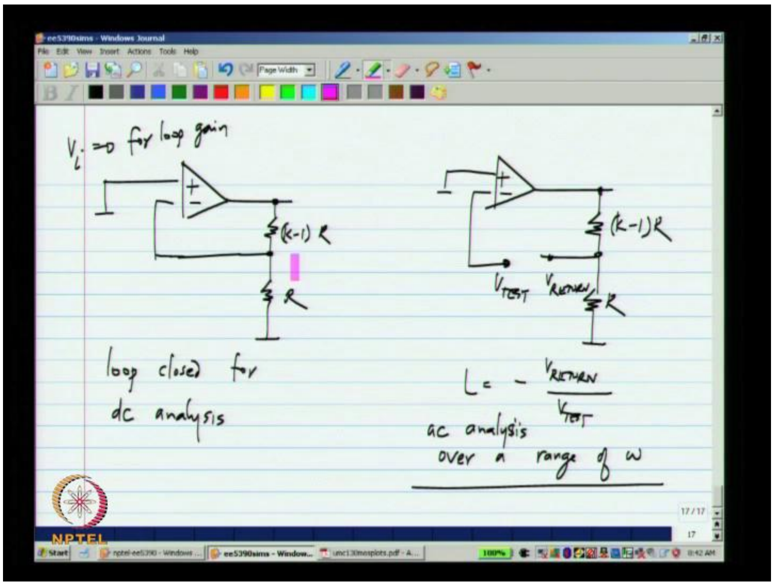


Let me take simple examples of an op amp amplifier, I will take a prototype non-inverting amplifier, I will also show the power supply in this case; obviously, when you use a simulator you also have to specify all the power supplies. Now, from here onwards I will omit the power supply, but it is assumed that you have connected the power supply in the simulator. Now, let us say I have to simulate the loop gain, the reason to do that is I have to ascertain stability phase margin etcetera, and adjust my op amp parameters accordingly.

So, what is loop gain, I break the loop here, I apply some v test, and I find out v return, I plot the magnitude and phase of v return by v test, which is the negative of the loop gain, and from that I can ascertain the phase margin and the stability characteristics. Now, this is fine, but how do we do this in practice, because this needs small signal incremental analysis like, I said you determine transfer functions using a c analysis, that is what we use for this particular case.

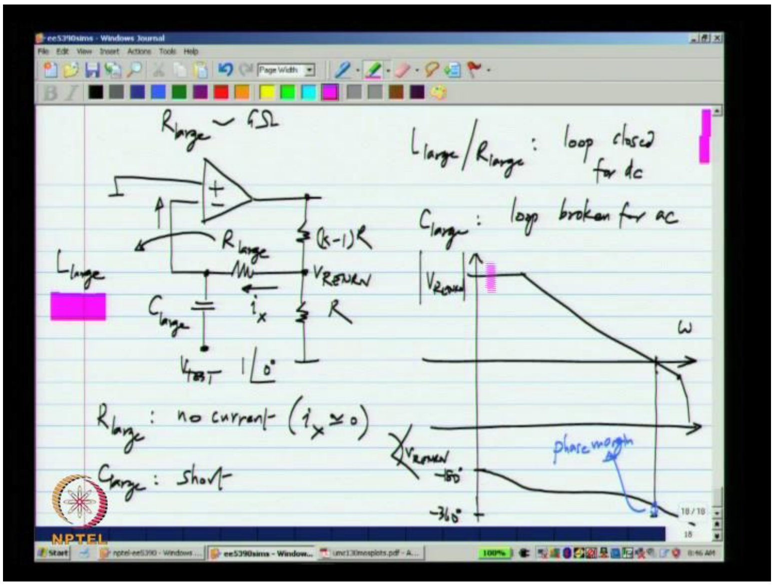
To determine the loop gain over frequency, but if we break the feedback loop the DC operating point is also destroyed, we know that op amp circuits in general negative feedback circuits, need negative feedback not only for operation, but also to setup the DC operating point. So, we must find a way that the operating point is not disturbed, but we are able to break the loop in some way, and determine the small signal incremental gain or mall signal incremental function in order to determine the loop gain. And when I compute loop gains, I set v i to 0, that is what I will show here onwards.

(Refer Slide Time: 40:21)



So, the circuits should look like this loop must be closed for DC analysis, but it must be open for a c analysis, and the loop gain is nothing but, the negative of v return by v test, and this has to be done for a range of frequencies. You do a c analysis over a range of omega, so how do we adjust this now we have done things like this before this is not difficult. Because, we have had cases, where you have to a c couple signals; that means, that the signal is applied for a c, but not for DC and so on, we will use a similar principle here.

(Refer Slide Time: 42:12)



And what can be done is let us use a very large resistance, and because this is a simulator not something that you have to implement in reality, this large resistance can be very large could be giga ohms also. Although, you cannot implement these things, you can put them in the simulator, in the apply v test through a very large capacitor, it again can be unrealistically large, it can have farads an kilo farads and so on.

The bottom line is that r large should be, so large, that no current flows that way and c large is, so large that whatever voltage, that you apply here will up here, there and as usual instead of large resistance, you can use a large inductor. Now, assuming that you have mos transistor circuit, the current here usually is 0, in that case you can use the resistance. If the current here is not 0, so you use a bipolar transistor or some other mos circuit or a current in this branch, where you have broken the feedback loop is not 0.

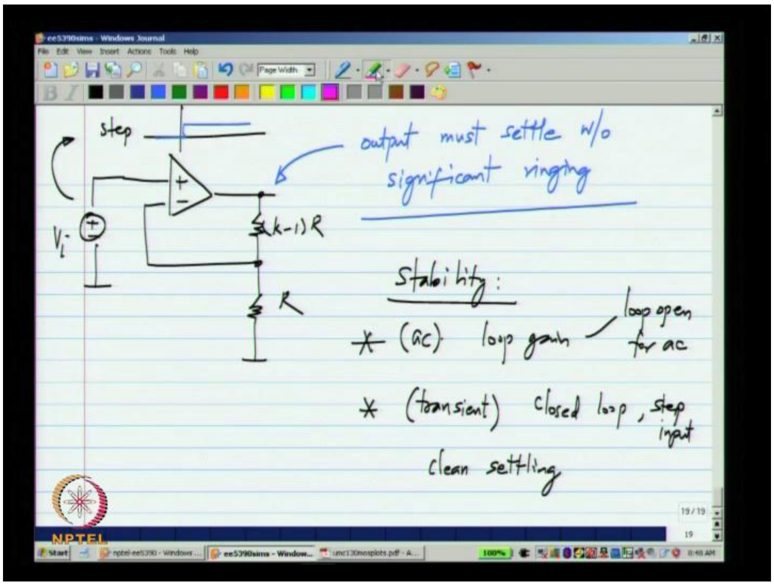
To use a large inductance instead, because then there is no drop across the inductor at DC, even if there a current flowing through it, so you could use resistance or inductance. The idea is to have the loop closed for d c, this makes sure that the loop is closed for DC, and c large which appears at some frequency beyond 0, will make sure that loop is broken for a c.

Now, because you can use the c large and r large which are unrealistically large values, this frequency limit can be very low, even be hertz or milli hertz, but once you have this arrangement, you simply apply v test here. You plot v return, as usual you apply v test over 1 angle 0, you get some v return, and you can plot the magnitude of v return, you expect it to be something like that versus omega, and you plot the phase of v return. Now, because v return by v test is the negative of the loop gain, this typically starts from the phase angle of minus 180 degrees, and it goes to 270 and then it goes off somewhere.

Let us say this is 360 degrees, you can find where the magnitude becomes unity, and the phase at that point and the difference form, this 360 degrees is the phase margin. The reason that it starts from 180 degrees is plotting is v return not minus v return, if you plot minus v return it will start from 0, and you should find the difference between the phase angle at unity gain frequency, and minus 180 degrees.

That gives you the phase margin and from this you can ascertain that the circuit has enough phase margin, and more importantly, what you can do is you can plot the magnitude and phase at different stages. So, inside the op amp there might be 2 or 3 stages, you can plot the magnitude and phase and you can also judge, which phase is contributing predominantly to the magnitude role off and the phase lag, and adjust the phase parameters accordingly. So, this will help you design the feedback system, so that it is stable, but finally, in any negative feedback system, you have to make sure that.

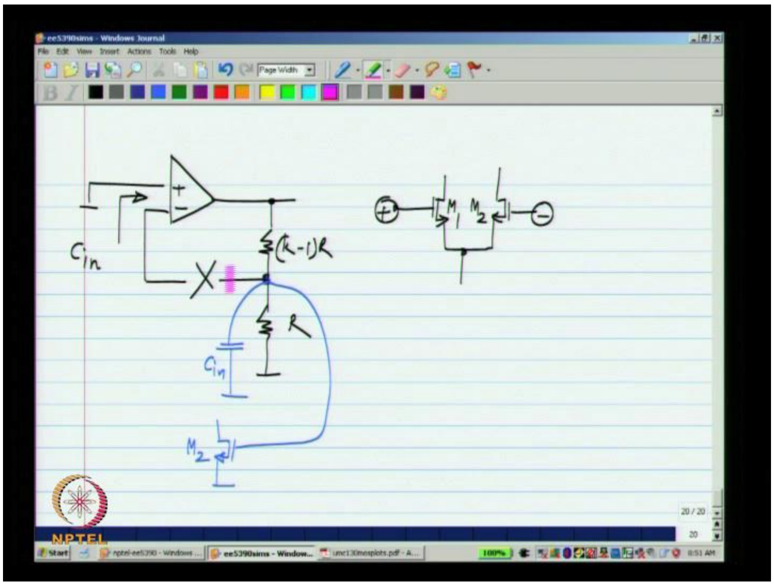
(Refer Slide Time: 47:01)



Once, the loop is closed the circuit behaves properly, that is you do a transient analysis let us say you apply a step here, you could apply a small signal step, and make sure that the output settles without significant ringing. So, basically to ascertain stability, what you do is you do an a c analysis for a loop gain with the loop broken, but the loop must be closed for d c, so that the operating point must be established correctly. And you do a transient with closed loop, and usually a step input and make sure that the settling is clean, you have to do both of these, because the transient analysis is the real analysis, that is what happens in a real circuit.

But, sometimes it is difficult to gage what parameter of the circuit to change, if the transient response has a lot of ringing, so you do the transient analysis, you get a lot of ringing. It may not be very easy to tell, which part of the circuit to change to fix, this it is easier if you do a loop gain analysis, and also find the n magnitude, in phase through different stages of the circuit. Then you will know which part to change finally, there is one small detail that I omitted while discussing this.

(Refer Slide Time: 49:11)



Whenever, you beak a loop, wherever it is you have be very careful about loading, I assume that I am breaking at the input of the mos op amp, so DC of course, there is no loading, but looking into here there will be certain input capacitance. When you break the loop, this node is no longer loaded by input capacitance, now this is a very important thing in loop gain, because if you have this loaded by the input capacitance. There will be some space shift from this point to that point, otherwise there would not be anything.

So, what you also have to do is to estimate the input capacitance, and put it there what can be done, for instance is let us say this is a conventional op amp with a differential pair, m 1 and m 2, which are identical. What you can do to mimic the loading approximately is simply connect, m 2 that is to this node you connect m 2 and leave it there. Now, this is only approximate, because what you see between the differential terminals is half the CGS of m 1 and m 2, and here we are doing the CGS of m 1 and m 2, but also this will not be in the saturation region and so on.

So, you can probably adjust the value of this one, but this will give you an estimate of the loading, due to another input this is again very important, because if you simply break the loop. And not take care of the loading, you could be missing out important space shift in the circuit, now one last thing in modern simulators, because this loop gain analysis is used very often.

You do not necessarily have to put a large resistor and capacitor to do this, I suggest that you do that anyway to get some practice, and better understanding of the loop gain and concepts like that, but in modern circuit simulators, there are also facilities to for instance to use a switch. Now, this switch is again not a real switch, it is a real switch for analysis, you can specify that the switch is closed for d c operating point, and open for a c. That is exactly what you want for d c operating point, the route must be closed, so that the operating point is established and for a c analysis it is open.

And this and there are also other more sophisticated ways of computing loop gain, you can consult simulator’s manual for this, but the principle is exactly the same, in the loop has to be closed for d c operating point, but be open for a c analysis. So, that you can compute the loop gain, now one of the things with the simulators is that initially you have to have enough practice, so that the simulator behaves as you expect.

So, the only way to ensure that is by practicing with many simple circuits, for which you already know the answer. Please do that you can try it with both the linear circuits and simple non-linear circuits, and then after that you can go on to more complicated circuits, such as op amps for which the simulator is indispensible.

Thank you, I will see you in the next lecture.