# CMOS Digital VLSI Design

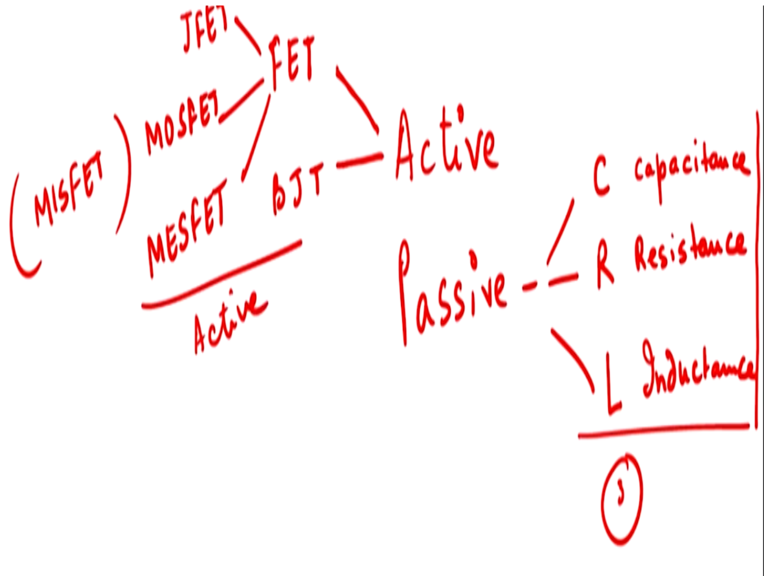
**Prof. Sudeb Dasgupta**
**Department of Electronics and Communication Engineering**
**Indian Institute of Technology – Roorkee**

**Module No # 02**
**Lecture No # 10**
**SPICE SIMULATION – I**

Hello everybody once again welcome to the new module of CMOS digital VLSI design under NPTEL online certification course this module is dedicated to the concept of spice and how to do simulations with respect to spice which means that spice is basically circuit simulator which will be primarily useful for determining currents voltages in a circuit. In this what we will try to do is something like this the outline will be introduced to you and then since it is more of a simulation based platform we will try to find out various example methods by which we can actually help you to simulate.

Why this why we thought of introducing this topic to you the reason being most of the digital VLSI design course generally you need to find out currents and voltages because of active devices at internal nodes of a circuitry now that can be done manually as well applying basic KVL and KCL but those will be only useful provided you have less number of activity devices and passive elements right. So let me define to you what are active and passive devices active elements. So the active elements which are available to you are primarily something like this that you will have.

**(Refer Slide Time: 01:53)**




What is an active element right? We have an active elements here and we have passive element. The passive element which is available to us primarily are there are three types of passive elements mostly used in electrical circuitry we have capacitances, we have resistances and we have inductances. So we have capacitances here right and we have resistances right and we have also inductances.

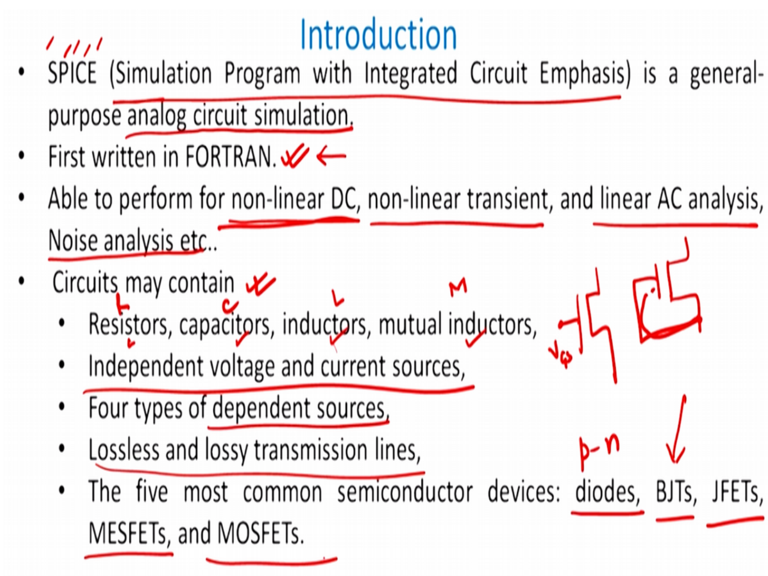
Now this as the name suggest most of the spice packages are most of the spice programs assume this to be passive devices they all passive devices in the sense that though the dissipate power but they do not have the active power source behind it where as active devices have actually power sources behind it so you require a power source to drive an active device.

For example you require for example of FET device right a field effect transistor can be a JFET or a junction field effect transistor or can be a MOSFET which is metal oxide field effect transistor or can be a MESFET metal insulator metal semiconductor field effect transistor right. This also name of this is MISFET metal insulator field effect transistor right so you can have active device like this with in this active device unlike FET you can also have a BJT bipolar junction transistor available to you and so on and hence so forth.

So what is spice does is that it tries to see these structures or these elements as either passive or active then given instruction by the user tries to find out independent voltages and current flowing these through individuals nodes and these elements right. Now to do that it does an internally mathematical manipulation this course will not be going into those areas there is dedicated course from that which is basically known as modeling through spice and you can consult that course if you want to.

But primarily this course is just to give an idea that if I am giving the spice package we can easily download it from the internet how can you simulate a simple for example simple RC element design or a simple MOSFET driving design. For example even an inverter which you have just now understood in your previous lectures. So to understand that this is we have got this three therefore recapitulate these three passive elements and you have got these three active elements available to us right.

**(Refer Slide Time: 04:22)**

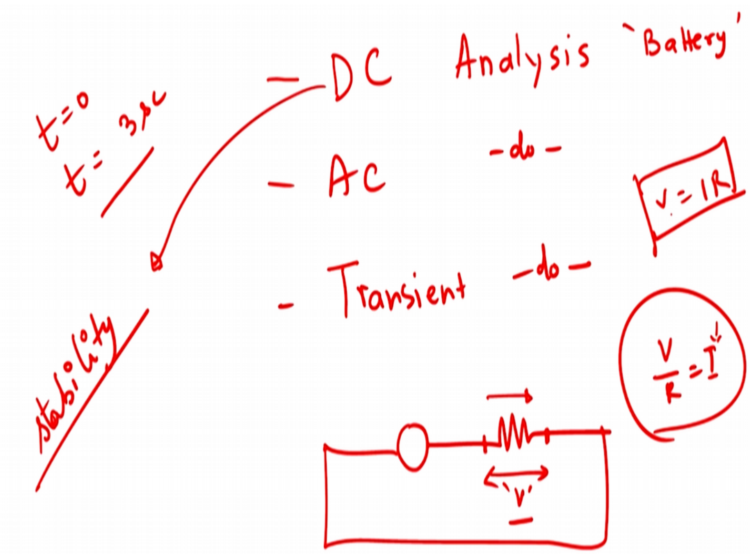



Now these are all open source and open source code available so let us go to the next slide and see how it works out what are the various activities available here. What is the meaning of spice? If you can see here spice means it is a the actual name spice stands for simulation program with integrated circuit emphasis right. So as the name suggest it takes an IC and then tries to run certain programs use based on certain models to find out the currents and voltages in general right.

It also used though it also used for analog simulation purposes you can also used for digital simulation purposes wherein you give DC bias and find out the voltage and current. It was initially when the spice was first used it was initially returned in FORTAN actually right but various version has come out with higher languages coming into this picture but spice when it is first introduced was FORTAN was able to do as we will see as of today spice is able to handle large amount of simulation right and what are the various large amount of simulation.

We have a non-linear DC simulation and explain these terms we have non-linear transient, linear AC analysis and noise analysis right. So I will explain to you what are the various analysis available to us. See types of analysis is spice does or for that matter anyone of us do while doing it in pen and paper.

**(Refer Slide Time: 05:53)**



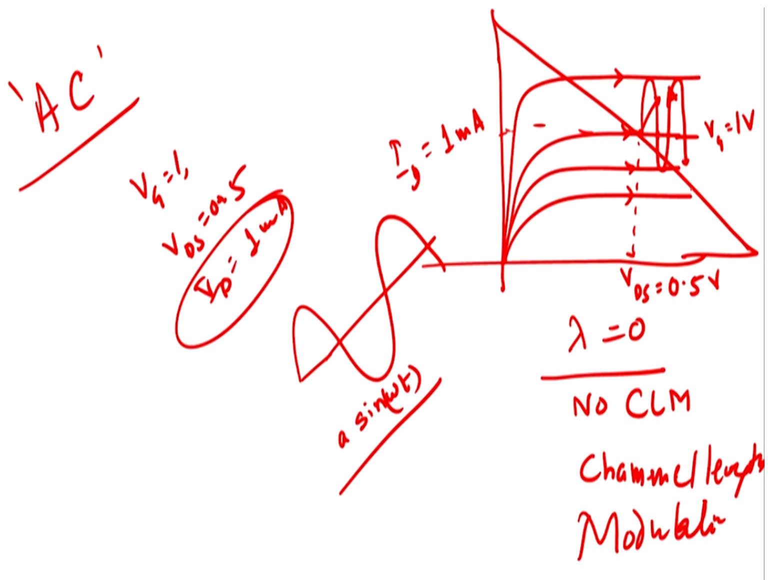

There are three actually analysis one is known as DC analysis right DC analysis you have got as I discussed with you I have got AC analysis and we have where transient analysis I think I have discussed with you all these in detailed in the previous module so in previous lectures but I have still like to give you a brief idea about what these are DC analysis as a name states that you apply a DC bias typically a battery source so you will have a battery or a DC source right and switch on it at T = 0 but you do not try to measure at T = 0 you measure at T = say any well say T = 3 seconds.

Where the voltages are actually stabilized across the circuitry so this DC analysis here relies on the fact that you will have stability in the circuitry which primarily means that the voltages of currents are not stable and now you can do a measurement right that is very important for you to understand right.

If you have a voltage source here and if you have a resistance here right I switch on the voltage source and then I see try to find out the current through it but then the current will not be instantaneous raise they will be a finite rise time. But after the current is (()) (07:12) it will latch on to its fixed value depending on the OM's Law as we all know V / R = I and therefore I and therefore when this current is reached we do the measurement try to find out that is the basic current this is known as DC analysis right or let us look in the other way out.

If a current is flowing through a resistor it is varying and at any one point of time the current is getting constant and it is not changing with respect to time then the voltage drop across the resistance is always fixed and that voltage drop is you can easily accommodate by finding out the value of I into R right. So these two basic voltage and current concepts should be very clear to us DC analysis concerned.

**(Refer Slide Time: 07:57)**




Let us look at what is therefore what is AC analysis right AC analysis is relatively difficult but we should know what is AC analysis is all about. DC means that you apply a DC bias for example a battery source or for example a source for example a MOSFET in its saturated state we remember we were discussing in a previous discussion but in a saturated state if I got a MOSFET without CLM which means by channel length modulation parameter is 0 no CLM right channel length.

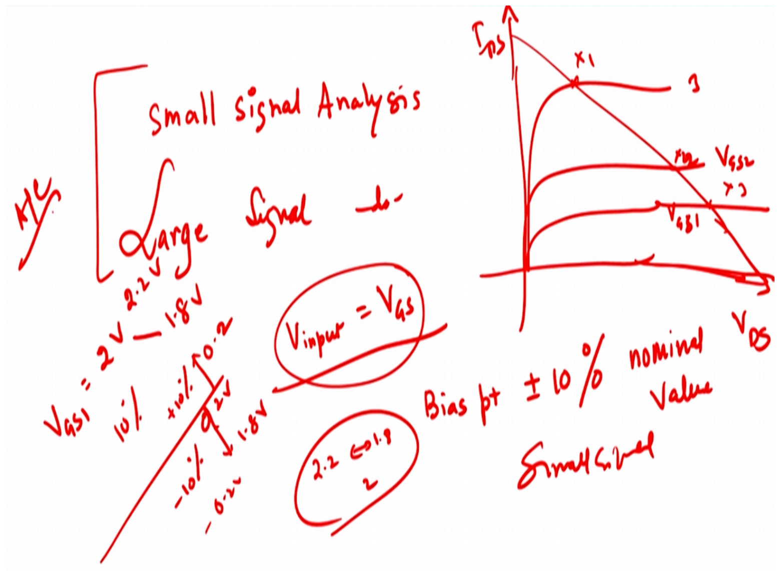
So there is no channel length modulation right if there is no channel length modulation with you. You can easily see that these are straight lines which means that if you bias your device and put a load line across this network and if you bias your device somewhere here with VDS equals to say 0.5 volts and ID equals to say I milli amp then we can say if we assume that if I put suppose this is my VG = 1 volt then if I put biasing criteria VG = 1  VDS = 0.5 right and ID = 1 milli amp I will always get a current source of 1 milli and it remains stable.

So this is a DC analysis which we do but let us do one more thing that once we have biased our device at this point let us suppose at point A we have biased the device. Now at the gate side of the device it start to give AC signal which is something like this whose value is given by a sin omega t then you can safely assume that this bias point will start moving up and down and then again up and then again down and so and hence so forth right.

When it goes up it gate voltage increases the current increases as it comes down it again becomes the original value and again it goes down to lower value of gate voltage current and then it again goes up and goes down. Which means that current is now therefore a function of time right as voltage was the function of time the current is also a function of time and therefore the analysis are that point of time in which the current or voltages are direct functions of time is basically defined as the AC analysis.

So what is DC? DC analysis where in you try to fix the value of the applied bias and then when things are stable try to find out the current or voltage at any of the point to device what is that AC analysis on the other hand is one in which you try to find out the value of voltage and current given the fact that the input voltage is a time varying voltage available to you and therefore if i want to find out the current which is varying and the varying as respect to time it defined to be AC analysis. This is AC and DC but we also understand what is small signal and large signal analysis in the spice simulation.

**(Refer Slide Time: 10:56)**




So let me explain to you what is small signal so let me explain to you what is small signal analysis right and what is basically a large signal analysis. Let me look at a small signal analysis going back to a previous discussion if you see a I can draw it as well. If you look at the IV characteristic of a MOS device and this to be a load line which this is the VDS this is IDS right and you have got VGS1, 2, 3 and so and hence so forth and it cracks at X1, X2 and then its X3 right and then it goes on to doing like that.

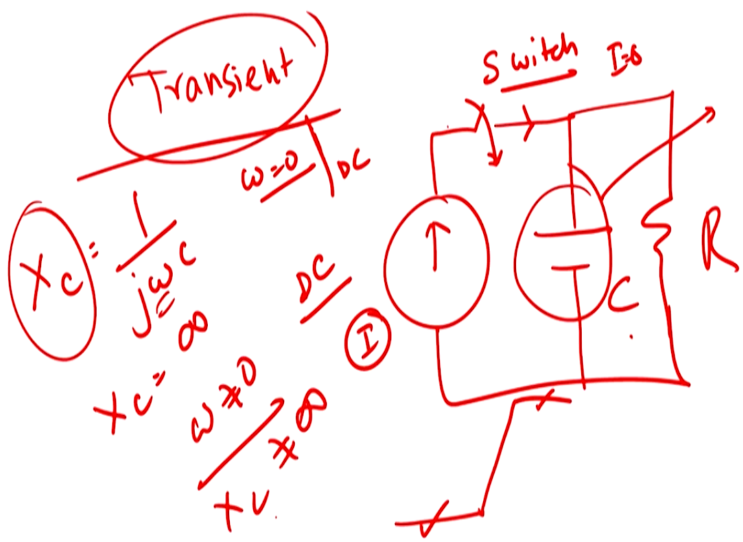
Small signal primarily means that when the input signals which is the input signal in this case? For all our practical purposes your input signal is basically V input V input is basically a VGS way to source voltages right and where you are applying the bias external bias by applying VDS. So the external bias for all practical purposes is VDS and the input signal is primarily VGS now if this input signal is VGS then we define small signal as.

If the bias point movement right so bias point movement is + - 10 % of its nominal value then we define nominal value then we define this to be as a small signal. I will give an numerical example to that for example if your VGS1 right say your where this is cutting this VGS2 sorry is say let us suppose 2 volts right and I have applied a signal which is peak to peak of say from 0.2 to – 0.2 then that these two volts will look as 2.2 to approximately 1.8 volts right which is 10 % of.

So from 0 value of 2 volts it goes to 0.2 volts which is 10% + 10% right and then it goes to – 10% is in the negative dimensions and you get 1.8 here with the -1 02 volts right. So therefore if that MOSFETS C is an input signal which is varying peak to peak from 2.2 to 1.8 with the mean of 2 anything less than that will be small signal anything larger than that will be as referred to as large signal analysis.

We will be seeing it later on or even subsequent modules that what are the possible therefore changes you want to do on the circuitry when you do a small signal analysis right. But as of today small signal and large signal is clear to you we also understood AC analysis DC analysis in transient so I can have a small signal obviously these are all applied to AC analysis for practical purposes right. So I have DC I have two types of AC small and large let me come to the last one which is basically a transient right.

**(Refer Slide Time: 14:02)**




I have a transient analysis transient is very simple and straight forward and explain to you what do you mean by that. Let us suppose I have a MOSFET here which is acting as a current source I also have a capacitor here I also have a resistor here right but this MOSFET it would have been a DC bias I would have try to find out the DC analysis then you will very much understand that impedance offered by capacitors 1 / J omega C with omega = 0 for DC analysis for DC XC will be infinitely large which primarily means that this capacitor will start behaving exact like a open circuit right.

So it will behave like a open circuit as if a capacitor does not resisting at all so DC analysis with capacitor in parallel or in CD's will not will parallel might wok but in series it surely not work because it will break the circuitry but let us look at another important issue here. The important issue is that no I do not do a I do not do a DC analysis here but I do transient means that I apply now a small switch here right a small switch here apply a small switch here I will explain to you what is small switch is how it looks like.

So what I do is I apply a small switch here is I am apply a this is S switch right so it can be ON and switched off. So let us about this is half so no current is flowing goes to 0 and current equals to 0 as you just press the switch right there will be a finite duration till which the current flowing from this current source will start to raise why do you start raise to finite time? Because you have a capacitance resistive element which does not let an instantaneous current to increase or charge to increase.

So you have to give a finite amount of time for the charge to sit onto the capacitor and as a result if you look at the input at this point if you try to find out the how the current is raising it is something like this. I agree as I discussed with you just now when at this point or at this point you will obviously have that capacitor open source and therefore automatically Ohm's law will be applicable but really what is happening now is something like this that you have close the switch but you are giving a finite raise time and therefore within that finite raise time my capacitor will behave like a impedance will behave like a impedance.

But it is value not will be will not be infinite because omega not equals to 0 so omega is not known not equals to 0 therefore XE will be not equals to infinite and therefore you will get a transient analysis (()) (16:44). So what is the transient analysis it means that for a very small duration of time if the voltages are rising or current are raising or voltages are falling currents are falling that how does the circuit behave under such a criteria is defined as my transient analysis.

So we have understood three types of analysis DC, AC and transient within AC we have small signal and large signal within transient we have we can have large number of changes or large number of issues we will coming into picture. Now as I discussed with you circuits therefore might have resistors might have capacitors might have inductors as well as virtual inductors we defined this as L we defined this as M, C and R right.

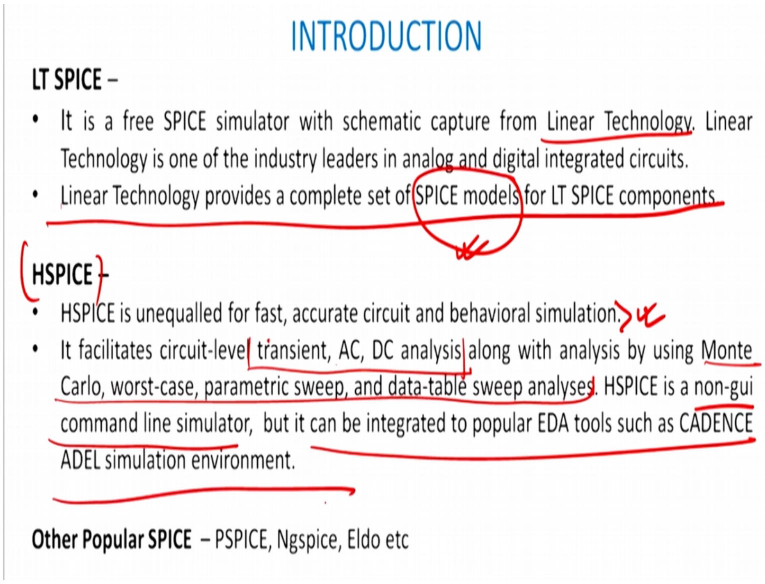
You can have independent voltage sources or current sources right we can have independent voltage or current we can also have dependent sources right this called interesting what is dependent source our dependent source in which gate and it is gate voltage for example an MOSFET gate voltage is controlled by the voltage available somewhere else which is not being under the control of the user right.

So for example I do something like this I have got a I have a MOSFET and this is just connected like this right then what happens is that whatever the value of voltage at this point is exactly goes to the value of voltage at this point and therefore you do not have a basic control over the gate voltage whereas what is an independent voltage independent will be something like this that you are able to give an external voltage which is actually being independent of the anything else.

You can actually giving give it from your side directly that is very important issue then then what we you also one more important point and that is lossy and transmission line concepts which means that we assume at this stage for your understanding purposes that for all practical purposes the interconnect will have 0 resistance 0 inductance 0 capacitance. So the basic model of spice assumes that it does not dissipate any power by virtue of switching and we also assume that resistance is 0 which means it is a perfect conductor and C is also equals to 0 that means it does not store any charge but obvious because it is a conductor right.

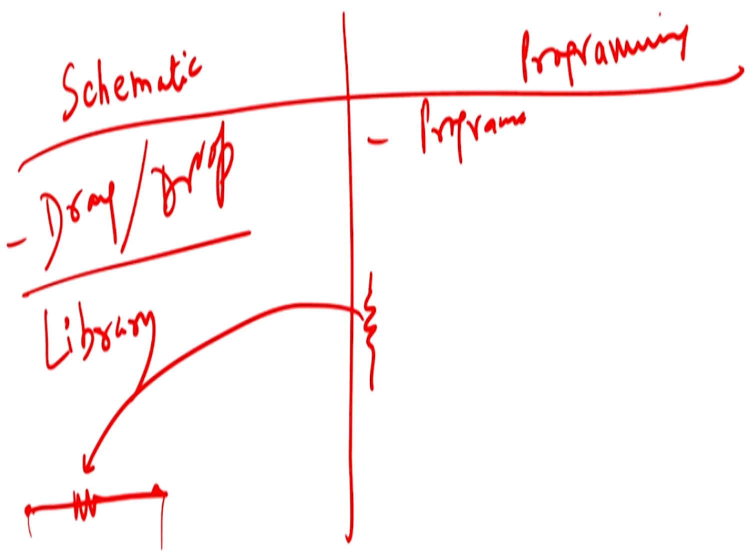
So these three assumptions are the basic assumptions which are important the five more important devices which we taken into consideration at diodes this is basically a pn junction diode we will have BJT bipolar junction transistor we have JFETs junction field effort transistor we have MESFETs which is metal semiconductor field effect transistor and we have MOSFETs which is metal oxide semiconductor field effect transistor. These all these three or five devices are equally important and have been used very often hand on this part what I would suggest to you at this stage is recommend is that go to the website internet and try to download LT spice.

**(Refer Slide Time: 19:39)**




LT spice is a free spice simulator and it is available from linear technology company is there linear technology right it is available there you can go to the website and they can download LT spice the free spice simulator and we have two types of options available there once is known as the schematic option another is programming option.

**(Refer Slide Time: 20:00)**



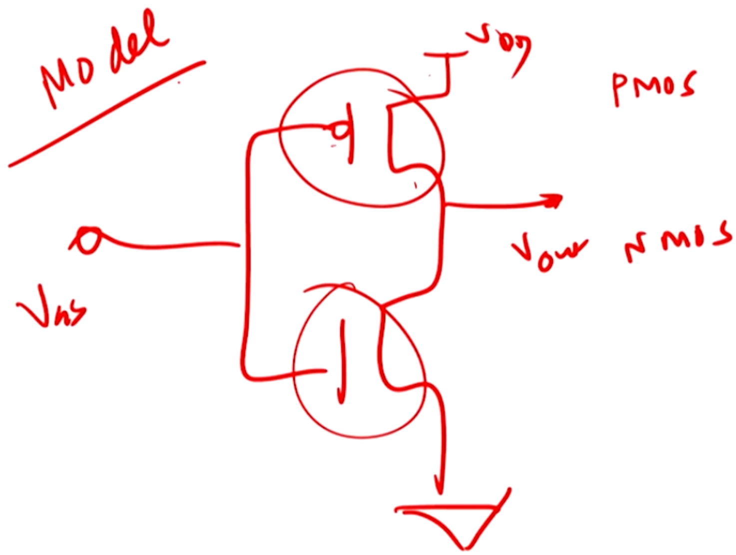

So I have got two option in it any other spice software we have schematic option and we can do a programming as well so we can do a programming as well as schematic. Schematic primarily means that you drag and drop so you have a drag and drop right where as here we have to write a program what is meaning of drag and dropping is? That you do a library available in your in your in your spice package from there you just need to click your mouse and bring any device or any passive device or active device from that point on to the schematic.

So you will have active any of schematic which is basically the area where you do the simulation and you will have an area where these MOSFET's MESFET's wires, capacitors inductors will be stored you just have to click there and drag it to the schematic and place it wherever you want to place. So let us suppose I want to have a resistor here right have a resistor here I want to connect to these two nodes I want to drag it here then turn it and simply connect these two together to form a simple resistive node right resistive node between this point and this point fine.

So difference between in the programming what you need to do is you to write a program physically as you do it in a other cases which means that I can show you the profile how you do it. But this is how you do it that means programming is relative if if not difficult but relatively slightly required certain amount of knowledge in terms of circuit dimensions various nodes available and so on and hence so forth.

That a schematic is more to do to C you are able to see it in front of you happening right and this is what the difference between the schematic and the programming all about. So what does it do is linear technology LT provides you the complete sets of spice model for LT spice simulation. Now you should understand very important model what is known as spice model what is the spice model we have discussed this point in earlier discussion also that let us suppose you have a MOS device right CMOS technology available to you.

**(Refer Slide Time: 22:08)**




CMOS please remember had a PMOS I a pull up case and you NMOS in a pull down case right if this is PMOS this is NMOS which is in front of you right and this is VDD and this is grounded and you apply a voltage here VGS and you get the V out here that replace this by a current source with overlap capacitance between gate and drain and gate and source also replace this NMOS by the current source as well as is respective capacitance is overlap capacitance is.

We will also have certain diode remember the back gate diodes right so whenever you are applying the second order back gate bias effects the PN junction had the reverse bias of the drain to bulk or source to bulk acts like a PN junction diode in reverse bias but if you make the voltage higher and higher there will be breakdown and therefore as you all of you are available in fact and therefore there will be current flow right.

That is simulated that is modeled as two diodes connected back to back and with the overlap capacitance available to us I have already defined that you are in the earlier case. So therefore we defined this to be as the model what is the model? Model basically means taking all the physical aspect of the device and putting it directly on to a schematic is defined as a model right so it take care of all the simulation phenomena is happening are all the physical phenomena it is happening within the device and tries to incorporate that into the circuit analysis here right.

So that is what the model file is all about in this case so when we say model we basically mean to say that the we are taking about that model which takes care of physics of the device. So that is S spice synopsis and it is very fast accurate in behavior simulation and it gives you a very good behavior which means that if you if you are doing a DC bias analysis and you are going to change the values of one and zeros particular node how does the output scenario changes will also be known to you not only that you will also able to extract the value of low to high propagation high to low propagation where at any intermediate between the circuitry right.

So that is the advantage of this spice model which you see in front of you it also allow you to do a very important property as I discussed with you transient AC, DC analysis we have already understood this it also answers to use in MC or Monte Carlo and parametric sweep and data tables analysis right. You do have to worry about these words do not worry at this stage but let me give you what do I mean by that.

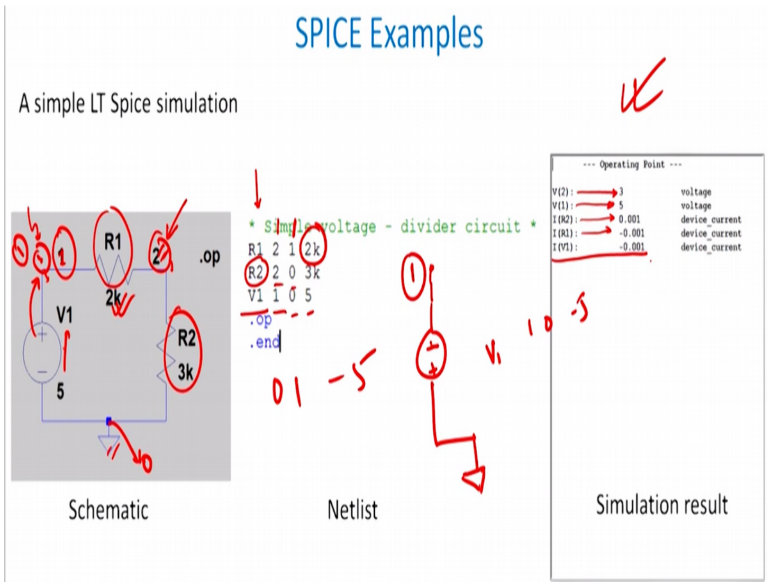
Say most of the device you fabricate in an industrial or in a FAB lab will automatically have a large amount of variation in its output characteristics you will ask me why and the reason being that for example in a MOSFET when you try to grow a 7 nano meter or 10 nano meter thick oxide layer right which is 100 angstrom you might end up actually having 90 angstrom or 110 angstrom right Cox changes.

As the Cox changes your ID changes so the problem is such small scale are but even in a small change in the input characteristics of the device will result or even the structural parameter of the device will result in the large change in the output voltage right relatively a large change in output voltage and therefore it is very important but we do a process variation analysis right which means that I vary I knowingly vary the oxide thickness by +-10% and check out how much amount of current I changing because of the change in the oxide thickness.

Now so therefore that is basically the sort of analysis which you try to do now Hspice is non-gui basically means that you do not have a graphic interface and therefore Hspice will not give you a exact graphic interface but less spice can actually write down the values of the system and you can actually predict the value of output current in output voltages but that advantages is such that it is comma line simulator and the very advantage of the it can be integrated with popular EDA tools such as CADENCE and ADEL simulation platforms that is the very advantage of the spice.

Though individually they are very good which is of course has been of use for quiet long time they get integrated also with the current CMOS technology simulators and they are primarily CEDENCE at this stage we can also do it with synopsis as well as with ADEL and so on and hence so forth. Others popular spice packages are PSPICE and NG spice, ELDO and so and hence so forth. So these are the various other spice packages are available which people tend to follow coming days right.

**(Refer Slide Time: 26:55)**




So let me come to a spice example now and let me show to you how was spice example works first of all you need to understand as a I suppose all of you are aware of a fact that when you are studying spice simulation you should know what is the meaning of node. Node as you already possible learnt in your B-Tech first year second year level courses that node is a element in the circuitry where two devices either active or passive meets with respect to each other.

So this if you look at the diagram here in this diagram then you can see that this R1 resistance R1 is connected between node 1 and node 2. Similarly this voltage source V1 is connected between node 1 and ground so this is a ground right and this is a ground. So what is the issue the issue is that we assume that we assume that the ground is one of the nodes at any voltage which we try to find out any other nodes is which respect to the ground unilaterally.

So when we try to find out the for example the voltage available at node 2 it is always this respect to the ground. Similarly at node if you want to find out it is always with respect to 1 so if you look at let us look at the way how to write it is that the first column here which you see is basically your the element. So I have a resistance here R1 which is between which two nodes 1 and 2 so R1 between 2 and 1 and the value is 2K so the value is 2K which you see in front of you.

Similarly R2 is another resistive element which is between 2 and 0 why because 2 is here 2 and 0 here and it is value = 3K right I also say that no there is a DC source which is kept between 1 and 0 whose values = 5. So we write V1, 1, 0, 5 and you can interchange 1, 0 to 0, 1 there is no problem on that but typically 1 to 0 means 1 to 0 primarily means if you convert from 0 to 1 it will be -5 right.

I can understand the reason why if it is 1 to 0 means 1 with respect to 0 right and that will be +5 volts why because the plus side of the DC source is connected to +1 right. If you just interchange it make it down I have got I have got negative and positive sign here and this is connected to ground here and this is connected to node 1 here. Then we define then we can define V 1 to be equals to be 1 and 0 with value = -5.

We do not give a common sign here I am sorry but what you try to do is you try to find out the value and it is give as -5. So you should be very aware of fact what is meaning of -5 value and why it is +5 or -5 DC current source right. Typical simulation where to see in front of you by your at the right place right and I just show it to you this is a typical results operating path. So you see V2 is 3 volts to the value of V2 is 3 volts the value of V1 is 5 volts the current flowing through the resistor R2 is 0.0001 ampere through R1 is R2 is 0.001 and through R1 it is -0.01 right.

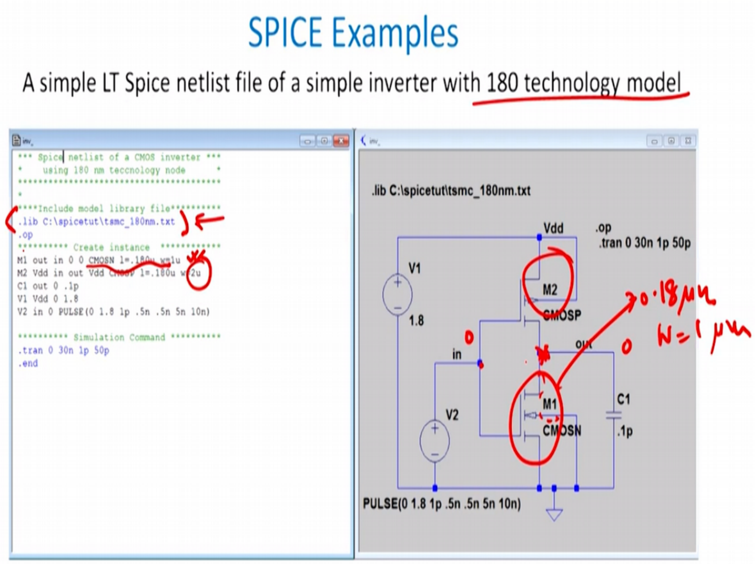
And the voltage across voltage V1 is basically the current flowing through the voltage source is actually goes to -0.001 ideally should be 0 right because voltage source is now any current to flow through it. But primary so I have a voltage source value = 1 right this is the general key of things now look at the general LT spice which has been made on the monte dynamic technology node when I say 180 nanometer technology I mean to say that go back to your previous discussion.

**(Refer Slide Time: 30:46)**


I mean to say that for a bulk MOSFET the distance between source and gate this is 180 dynameters right so when I say 180 dynameter technology I means to say that now I am using a device or a bulk device who is source to drain distances approximately equals to 180 dynameter as you can see here.

We also say that if this is 180 dynameer technology I primarily mean to say my W can be changed because W / L is changeable my W can change so that I get a minimum width design or I can get the width design right maximum width means if you increase W everything else remaining the same you will get a larger and larger current available to you at the cost of higher and higher silicon area which you pay as the cost. Keeping L constant of course minimum value which is 180 nanometer in this case.

**(Refer Side Time: 31:37)**




But this is how you define your over overall design in this case so I missing 180 nanometer technology model now you are asking what is this model well this is nothing but taking the bulk MOSFET at 180 Nanometer if you look at this point then if you look at this line of the spice dot lib c colon backslash spice tut so and hence so forth what is that tell means that I have some where a model file available and I am calling or invoking the model file available what to be schematic or want to my programing.

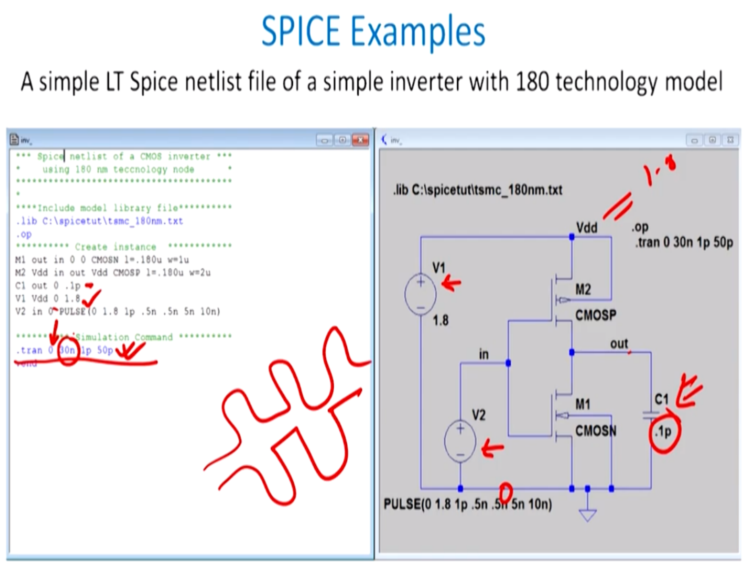
What is model file do? Model file will give you the value of all the parameter required to involve or to calculate the current or voltage at any intermediate point in the any circuit range right. So that is the reason we use a model file and for example if I look at this basically TSMC model file TSMC is basically Taiwan semiconductor manufacturing corporation and it as provided to the user 180 dynameter technology file which means that how does the MOSFET behave at 180 nanometer is already known to you in terms of the technology files.

You just have to plug it and place it where ever you want to do with and the job will be almost done for your case. So that is very easy and easy to do it and easily get back to the issue very well. So this is M1, M1 is between out and in M1 is this one right is between what? Between out and this is your input which you see this out and this is in here. So this is your M1 you kept here because this is grounded so this is the first table is nothing but M1 which is this one between out and in now initially held and 0, 0.

So out is held at 0 and input also held at 0, 0, 0 CMOS 1 = 0.18 so this is basically based on 0.18 microns technology right and it is W has been known to be 1 microns now what I do is as I discussed with your in our earlier discussions this is the mobility of holes is approximately 2 to 3 times smaller that of electrons  in order to sustain equal TPH and TLH I require that the width of the transistor and PMOS we made actually 2 to 3 times larger right because resistors are smaller.

So I want to make it larger so that it resistance actually again falls down to a value exactly goes to that of NMOS. So if you look at this PMOS which you see PMOS is typically which is M2 here PMOS has got typically 2 micron here width which means it is almost double of NMOS right. This will ensure to me that my low to high raise high to low raise is almost constant.

**(Refer Slide Time: 34:24)**




What is V11 what so what is now C1? C1 is this one of a award value 0.1 picofarad so the C the C1 is between out and 0 so it is between out and 0. So you see the default value is that you cannot have any of the nodes of spice or circuitry we have floating node it has to be either given to the ground low impedance part or to an input source you cannot float it once you float it simulation package will show an error to you right.

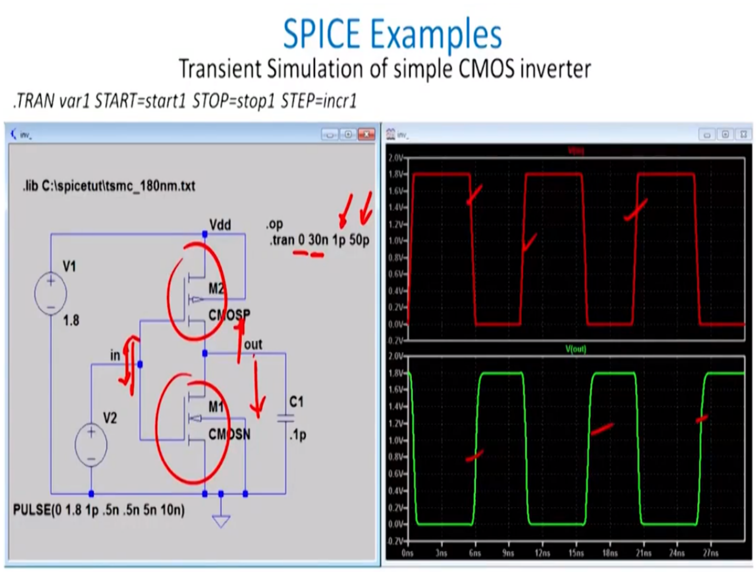
So it is pretty essential that you appreciate the point for all the input parameters or output parameters even active device or a passive element should be either grounded or attached to a appropriate device level. For example in this case C1 is connected between some ground and output V1 is connected between VDD and therefore you see between 0 and 1.8 volts so this is

1.8 volt and this is actually a 0 volt here you can also write down VDD and 0 volts here I can chase the value VDD later on similarly what is V2?

V2 is this value right but V2 is what V2 is basically a pulse signal is something like this pulse signal is given to you and therefore the output of the whole passing element will be will be something like this because it is a basically a static inverter which you see in front of you. And therefore there will be mutually complementary which respect to each other what I can trying to tell you is that for example if you look at this basic point here in this case. I said tran dot tran 0 30 1P 50P it means that I will do a transient analysis right starting from 0 right approximately 30 seconds with a margin or a estimation to be approximately 50 pico seconds right.

So at every 50 pico second I will try to mark what is the value of current voltage flowing through the circuitry right. So this is what you see in front of you and this is a command which you see in front of you right so this is the pulse mode command which is basically this this pulse mode command which you see in front of you.

**(Refer Slide Time: 36:26)**




We look at the transient simulation as I discussed with you in the previous case but this is my basically my input right this is my input and this is my output which you see in front of you and they are mutually like a when the input goes low output goes high and vice versa right when the V input goes low when this goes obviously this will go low this will high because it is charging to M2 and therefore this go high when this goes high right and this goes high NMOS starts to discharge and therefore this goes low.

And that is what you see in front of you the red ones are primarily the input and the green ones are output which is available or you must have appreciated by this type around that the change is on instantaneous in nature there is always a finite time duration between low and high states right. So please be very careful while designing circuitry or for understanding purposes that it is getting very difficult in order to make this circuit work as the PO switch or a PO device.

Because this is a finite rise and finite fall which is available to you this state and that is quiet critical quiet understandable. As you can see I go from o to 30 nano second and this transition period has got 50 1 pico second in the transition period I go up to 50 pico second just to find out the values of each individual value of voltages and current in this case right. So the increment values are given to be approximately goes to 50 pico second.

So I start with 1 pico second staring time from 0 to 30 1 pico second I try to find out the voltages and current and try to find out. Now another example is so the previous basically your transient simulation which means that you are using a simulation in which we are using transient analysis because you are just given a voltage rise or fall and then you try to find out the voltage or rise or small at any subsequent regions of operation.

Whereas what is happening here is that we have applied the DC sweep what is DC sweep? DC sweep is 1 in which you apply the voltage right you apply the voltage and as you go on applying voltage and carrying the voltages the current starts to show a rise and fall. So I will take up this one next module and till then I will explain to you from where I got this in next module. So by this module let me finish with two to three sentences that we tried to find out the value of current or voltage using three analysis transient, DC and AC analysis.

Within AC we will have small signal and large signal and within DC we will have either a fixed bias network available to me a small signal fixe d bias you could also have variable bias available to me but primarily we have only looking at how to make therefore of spice program available to me right. So in the next slide we will continue with the device simulator work and then we will go to combinational logic thank you very much.