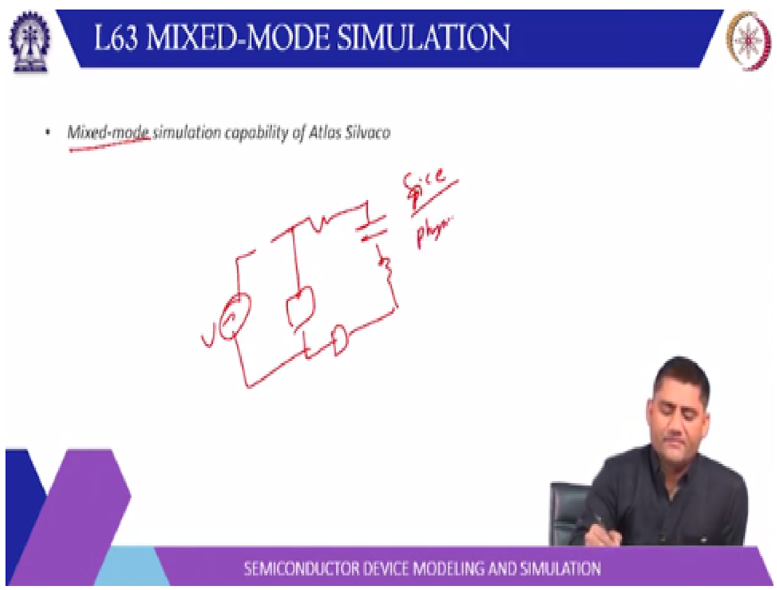
Semiconductor Device Modelling and Simulations
Prof. Vivek Dixit
Department of Electronics and Electrical Communication Engineering
Indian Institute of Technology, Kharagpur

Lecture - 63
Mixed-Mode Simulation

Hello, welcome to lecture number 63. This will be the last lecture and here I would like to inform you about a feature of that software or that is also a requirement that the device simulation itself is good for you know finding out the characteristic of the device and research purpose but the performance has to be checked in a circuit.

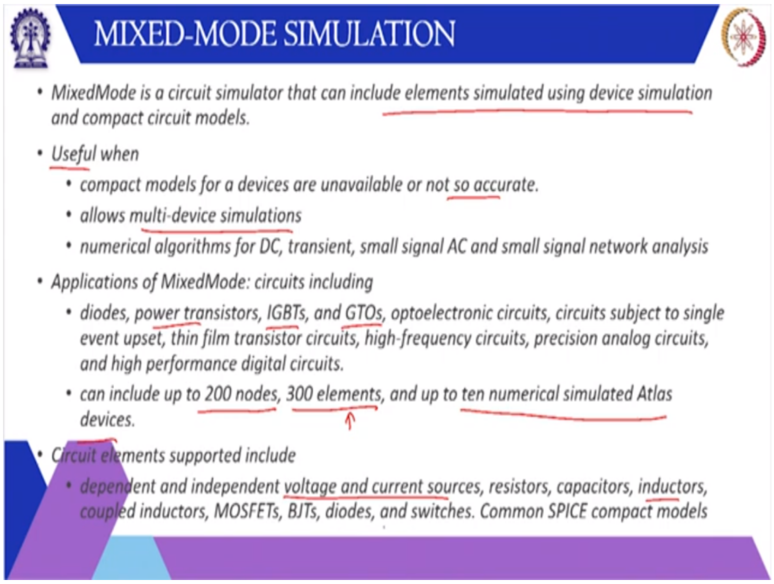
(Refer Slide Time: 00:57)



So, this Silvaco software allows us to allows us to do simulation called mixed mode simulation, so in mixed mode simulation what you can do you can have circuit framework let us say R C L whatever. Then in that you can put number of devices and simulated basically is with some excitation. So, this allows you to have a mixed mode simulation where you have rest of the circuit is mentioned specified through spice like program.

And then you have your physical device also here. Now this is useful in case you are not able to get a compact model for your device because all the **(()) (01:33)** simulation they use the compact models of the or you do not want to use a compact model you want to use a physical device and see the device performance. Then this mixed mode simulation is a good capability that is available with the Atlas Silvaco.

(Refer Slide Time: 01:46)

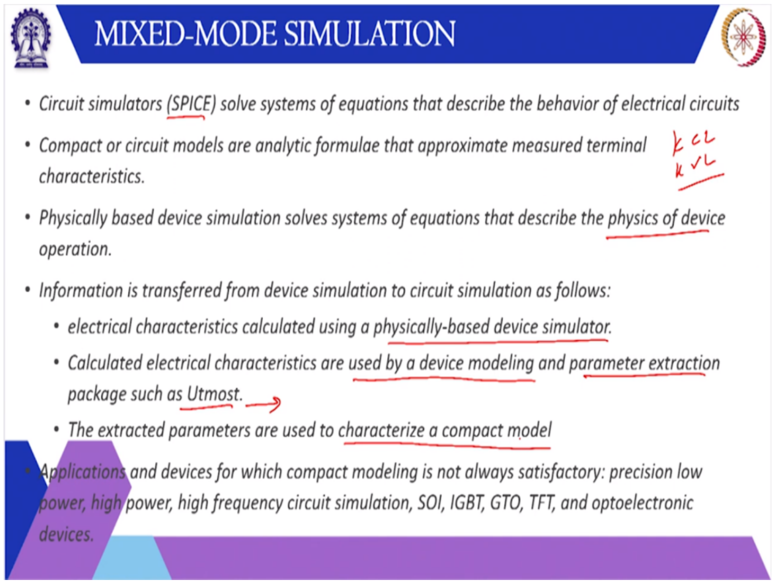


Now mixed mode is a circuit simulator that include the elements simulated using the device simulation and other elements can also included. So, you can have n number of such devices or you can use a compact circuit models can also be used. Now when it is useful compact models are useful and mix mode is used when compact models are not available or they are not so accurate.

Or it allows the multiple devices to be simulated together, numerical algorithm for DC transient small signal AC and a small signal network analysis. They are used to you know under calculate the effect of this mixed mode simulation and such devices like diode power transistors, insulated gate BTs great unknown optoelectronic circuit, you know single event upset, a thin transistor high frequency circuit, precision analogue circuits, digital circuit.

So, there this mix mode can be used and there is a limit on the mixed mode that it you can include up to 200 nodes up to 300 elements up to 10 numerically simulated Atlas devices. So, these 300 elements include the compact models and so on. Now circuit element that are supported you can have voltage and current sources dependent or independent sources also, resistors capacitors, inductors, coupled inductors, MOSFETS diodes switches you know **(()) (03:29)** and so on.

(Refer Slide Time: 03:31)

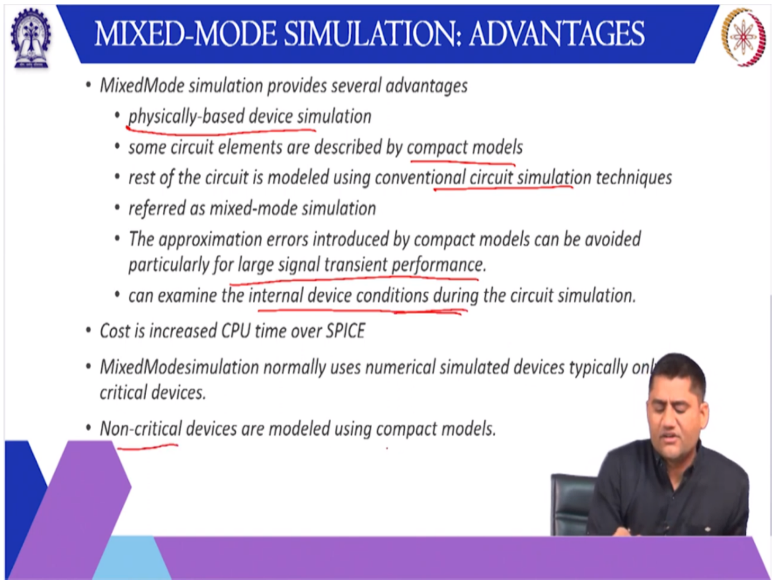


Now circuit simulator SPICE it will solve for the you know KCL and KVL equations for the circuit and this compact or circuit models are the analytic formula that approximate the measured terminal characteristic and physically based device is solved using go atlas and subsequent commands based on the fix of the device. Then this information from the devices transferred to the circuit simulation.

So, electrical characteristic calculated using physically device simulator and then this calculated characteristic are used by the device modelling and parameter extraction. Utmost basically is a tool which is used to extract the you know model parameters and these extracted parameters are used to characterize a compact model. Then application and devices for which compact model is not always satisfactory.

So, these are basically you know the high-power high frequency circuit simulation SOI, IGBT, GTO transistor optoelectronic devices, so in these scenarios we may have to use the mix mode simulation.

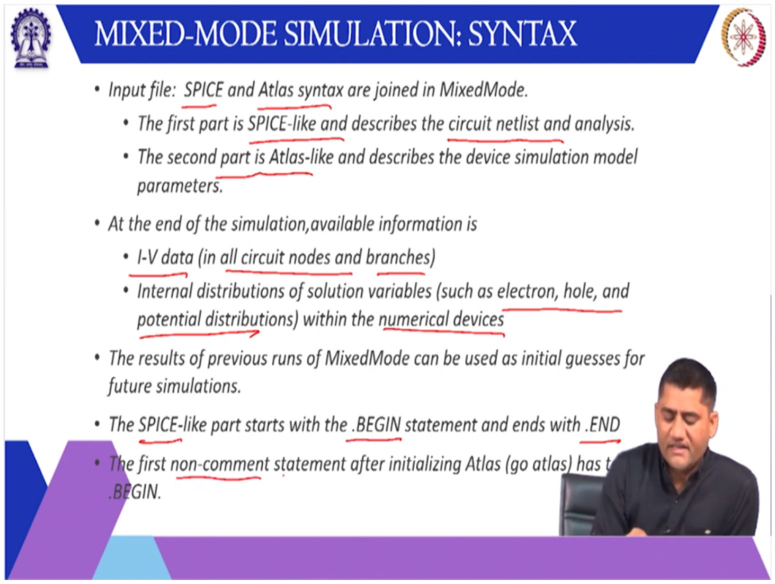
(Refer Slide Time: 04:41)



Now it also provides several advantage you can look inside the physical device in the circuit framework you know where what are the voltage values, what are the you know in different region, what is your generation you know, what are different critical parameters. And of course, some other circuit element can be described as a compact models and rest of the model is modelled using the conventional circuit elements, conventional circuit simulation techniques.

The approximation error introduced by the compact model can be avoided, especially for a large signal transient performance and we can also when the internal condition of the device the numerically simulated device. Now of course, it includes a increased CPU time but the information we can get is very useful and important and we only use the critical devices as numerically simulated devices. Non critical devices we instead of them, we can use the compact models.

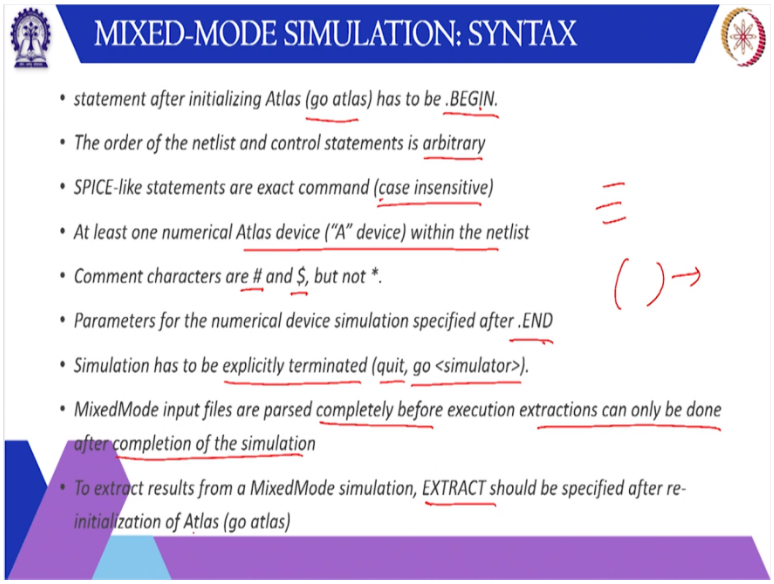
(Refer Slide Time: 05:50)



So, the input file will have both the syntax the SPICE like syntax and the Atlas syntax, they are joined in a mixed mode. So, the first part of the code is basically SPICE like syntax and it describes the circus netlist and analysis part. Second part is Atlas like command and describe the device simulation model. At the end of the simulation the information that is available is the IV characteristic at all circuit nodes and the branches.

And the internal distribution of the solution variables such as electron concentration, hole concentration, potential distribution, their velocity is band profile within the numerical device. The result of previous runs of mixed mode can be used initial guesses for the future simulation and regarding the syntax the SPICE like part start with the dot BEGIN statement and it ends with the dot END statement. So, the first non-comment statement after initializing Atlas has to be dot BEGIN.

(Refer Slide Time: 06:52)

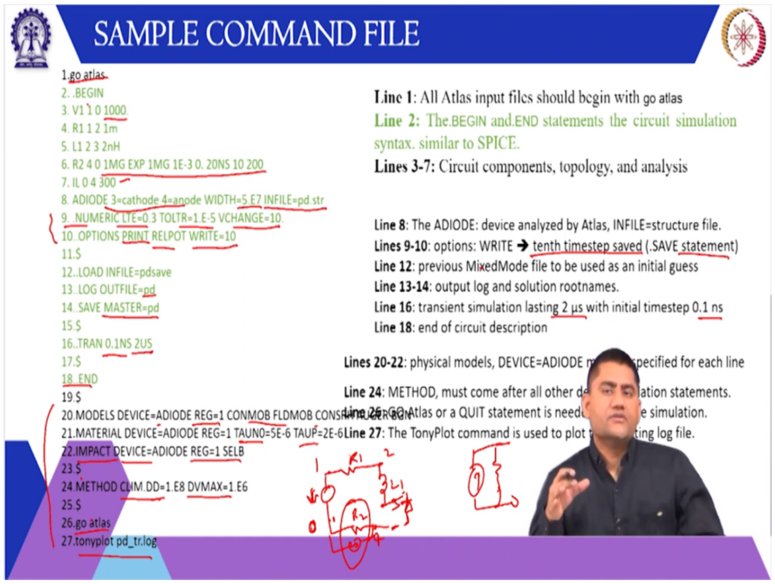


So, here you can see one example in the last next slide. So, after initializing the atlas go atlas, it has to be dot BEGIN. Then order of the net list and control statement is arbitrary so that is immaterial you know the way you want to define and SPICE syntax are case insensitive. So, at least one numerically simulated device has to be used in the netlist, then only it will work and the comment characters are hash is already comment characters but dollar is also a comment character here.

Parameter for numerical device simulation are specified after dot END. So, between the dot BEGIN and dot END is all spice like syntax. And then simulation has to be explicitly terminated like quit or go simulator go at loss and so on. Mixed mode input file they are passed completely before the execution unlike the normal Atlas file. So, there you know commands are executed line by line but in case of a spice it is compiled and then it is simulated.

So, that means the input file has to be completely passed before execution. Then extraction can only be done after completion of the simulation to extract the result from an export simulation extract command should be specified after re initialization of the atlas. We have to write go Atlas and then write the extract statements.

(Refer Slide Time: 08:21)



So, this is a sample code for mixed mode simulation, so you are invoking the Atlas simulator there is a go atlas, then dot BEGIN. So, there is a voltage source between node 1 and 0 and let us say this value is 1000. Then resistor one between node 1 and 2 let us say this is node 0 between node 1 this is 1 this is 0. So, there is a voltage source here then between 1 and 2 there is a resistor R1, this is the node 2. Then between 2 and 3 there is inductor, so this is L1, this is the node 3.

Then 4 and 0, they raise this is 1 0 1 2 2 3, then between 4 and 0. So, let me write like this 4 and 0, there is IL, so this between 4 and 0, there is R2 one MG exponential. So, some kind of between two and three there is inductor L1, between 4 and 0, there is R2. So, between this is zero terminal and 0 and 4 is R2, so this is R2 between 0 and 4 and there is a IL independent current source between 0 and 4. So, there is a current source also here, between 3 and 4, there is a diode.

So, here is a numerically simulated device, so 3 is cathode, 4 is anode. So, you have this device here connected to 3 and 4 here, then its width is mentioned because in 2D device simulation the width has to be mentioned otherwise you get the current density per unit length. So, this is pi V7 that is a width in Micron so that means it is 5 to 10 raised to power 7 Micron, so some you know. Then the N file is mentioned p d dot str does the structure file used in the simulation.

Then numerical low concentration error tolerance and voltage change they are you know not numeric commands are there then dot option print relative plot. These are the basic commands with respect to the 9 and 10, tenth time step save statement and then you lower it in file pd save that will basically it will load the in file then log out file is pd. So, the finally pd dot log and that same master means all the subsequent file have this pds common.

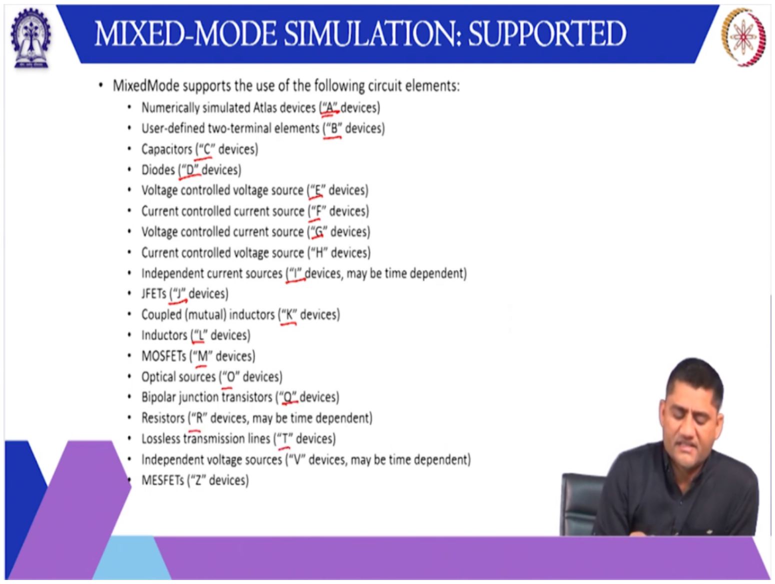
Then we are doing transient simulation for 0.1 nanosecond to 2 micro second so and dot N so that basically ends the spice like syntax. So, here it is described basically no, so a tangent simulation lasting for 2 micro second with the step size or the initial step size of 0.1 nanosecond. Then after that from 20 to 27 is basically the models physical models used by the device. So, structure file does not have these models so these models have to be specified in the command itself.

So, here model device is the a diode region one these are the model CONMOB FLOMOB and so on, then material TAUN TAUP, then you may have to this impact device region Salvaco model method shield limit of the this doping DVMAX. Then go Atlas and Tony plot, so this will end here and for **(()) (12:16)** again either go atlas. So, what this tip is basically doing here? There is a voltage source here which is fixed voltage DC voltage. So, it is some kind of bias basically.

So, it is probably you know biased this device PN junction diode here with a series resistance inductance and this is a current source with a parallel resistance, so that means we have included a realistic current source because real current source can be represented as a current source with some series parallel resistance or similarly a real voltage source can be some voltage source with a series resistance and so on.

Now what is done? This parallel resistance is modelled as variable resistance. So, 1 MG 2 you know 1 MG, it in some step size, so this basically simulate this the reverse recovery of this diode under the circuit framework. So, this is a very nice example to illustrate the use of mixed mode simulation in understanding how the diode responds in a circuit framework.

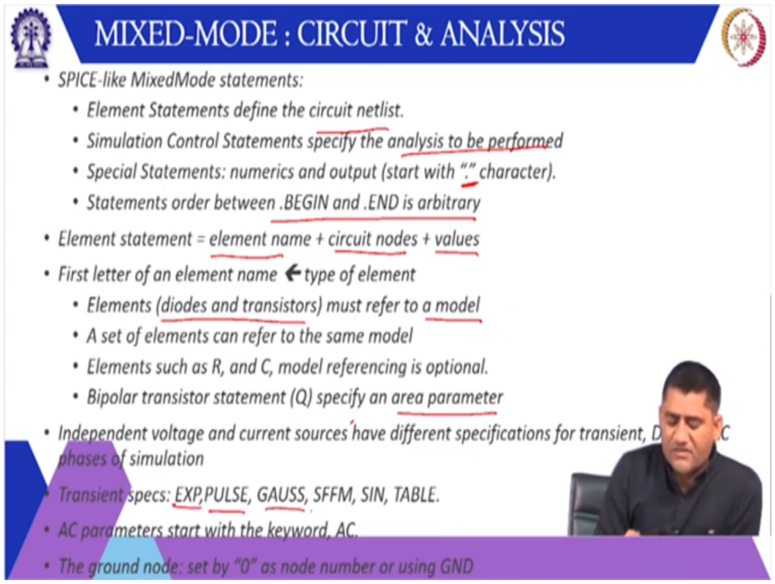
(Refer Slide Time: 13:32)



Now mixed mode also support the devices but there is a naming convention. So, for numerical simulated device the name has to start with A. So, that is why the name of this device is a diode. So, the name has to start with a there is a requirement for user defined two terminal elements you can have this B capacitors has to start with C, diode has to start with D, voltage control source has to be E, current source F, then voltage controls F, G, now independent current source is I.

So, you can see here this IR is the independent current source, then V is the independent voltage source, then in R l is the inductor R is a resistor and so on. If you want to include JFET the name has to start with J coupled mutual, inductor is to start with k, inductor L, MOSFETS, optical source with O, transistor BJT with Q resistor R transmission is T, voltage source with V, MESFETS with Z. So, this is dynamic convention that is followed by the mixed mode simulation.

(Refer Slide Time: 14:37)

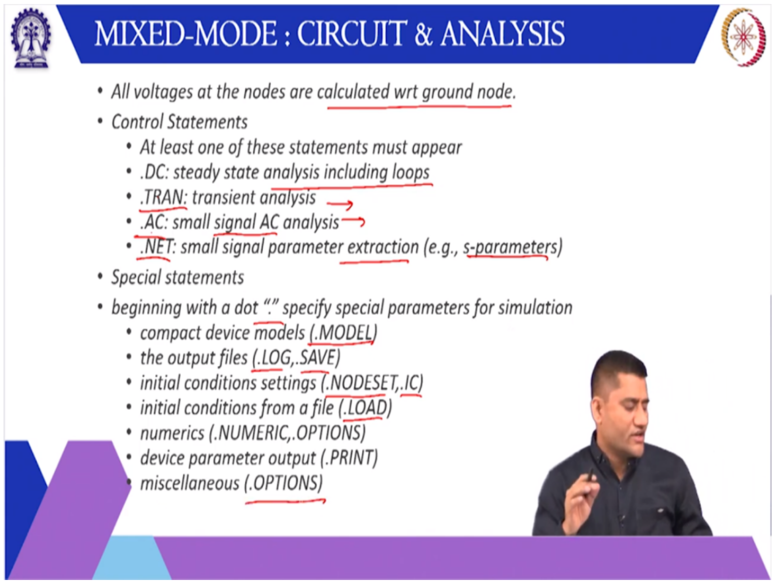


So, element statement define the circuit net list then simulation control statement space analysis that is to be performed. So, here is the dot transient and then a special statement like numeric or output they start with dot character. So, a statement between order between BEGIN and dot BEGIN and dot END is arbitrary and then whole thing is compiled and then simulated. Now element statement is the element name followed by the circuit nodes and the values associated.

Then first letter of element tells you the type of element and the elements like diode and transit they must refer to a model. So, either it is a physically simulated device or it is a device having a compact model. Now elements RC model differencing this is optional and for numerically simulated device also you have to specify the area parameters. Then independent voltage and current sources have different specification for DC AC and the phase of the simulation.

That for transient space you can use exponential pulse Gaussian and so on. An AC parameter is start with AC and the ground is set by zero so zero is by default is the ground.

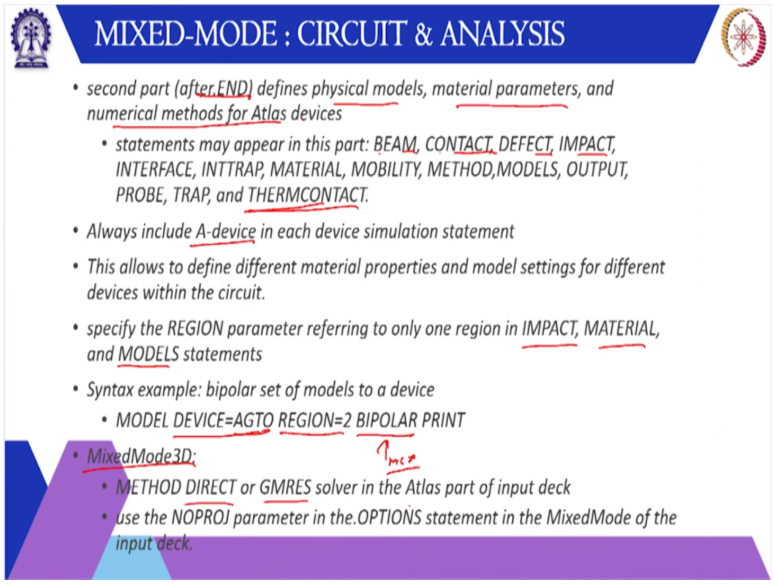
(Refer Slide Time: 16:04)



Then all the voltage of the node are calculated with respect to the ground node and the control statement dot DC basically it is a steady state analysis including loops, dot transient, dot AC small signal AC analysis. So, dot TRAN and dot AC, the difference is that it is a small signal and in transient you can use the larger change in the voltage or the control parameter and dot NET is basically extraction of small parameters like s parameters can be extracted using dot NET.

Then a special statement they begin with a dot. So, you have to especially a special parameter of simulation. So, like compact device model you include using dot model command output files dot LOG or dot SAVE initial condition dot NODESET dot IC. If you load from a file then dot LOAD numeric parameter device parameter miscellaneous, they can be a specified using these dot commands.

(Refer Slide Time: 17:03)



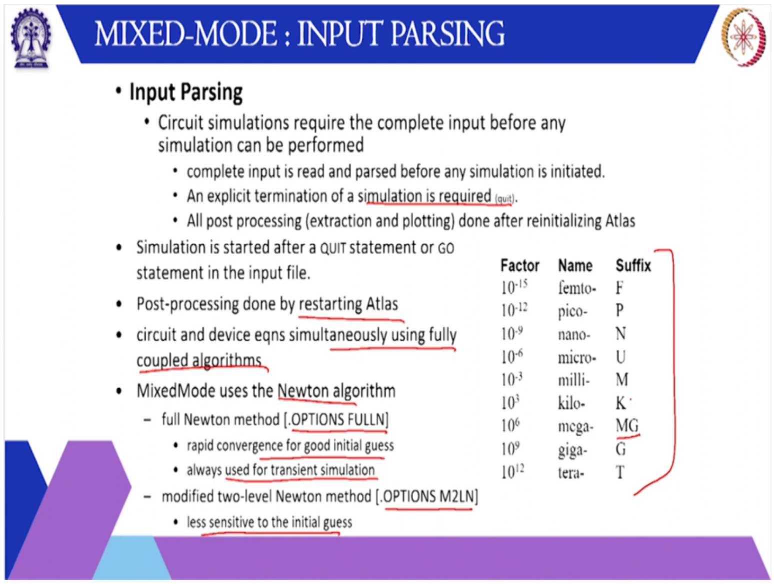
Now second part which is after dot END so that is basically Atlas physical device simulation, there you have to specify the physical models the material parameters and the numerical method for the physically simulated device. So, here there is a you know term like Beam for optical, contact type you know ohmic or set **(()) (17:26)** contact defects impact thermocontacts are basically used in the non-isothermal simulations.

So, this thermocontact what it does if you specify the thermocontact then the contact will basically have some thermal conducted will be defined along with this one using some alpha parameter. And the temperature of the lattice will vary with the you know electron temperature and so on. So, now this the normal contacts have to be specified as thermocontact also for non-isothermal simulations.

Then of course physically simulated device has to be included in each mixed mode simulation and then it allows the user to define different material properties and model sighting for different devices within the circuit. Then the region parameters have to be specified because the structure file does not have all the information. You have to especially these parameters like impact or material or model statement.

So, the syntax is like model device name like here is AGTO, so the name is AGTO because it is a physical simulator the region 2 is bipolar is a macro. So, it include all the models used in bipolar simulation print. You can also use mix mode 3D, so here the method can be direct or GMRES solver in the atlas part of the input deck. You can use no projection parameter in the dot OPTION statement.

(Refer Slide Time: 18:58)

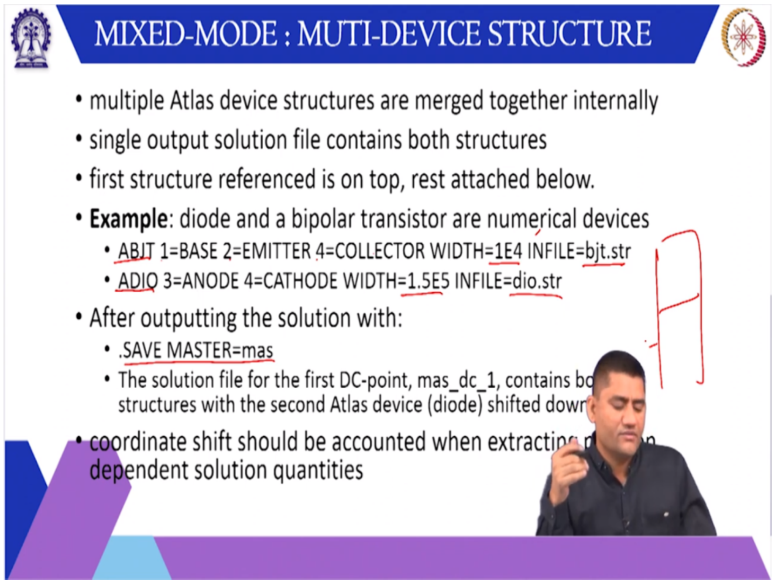


The input parsing the circuit simulation requires the complete input before any simulation can be performed. So, a whole input file is read and passed it basically this is compiled and then the simulation is initiated. Then explicit termination of the simulation is required that is quite or using go atlas. So, all post process will have to be done after reinitializing, so you see the tony plot was after you have to write go atlas. Then the tony plot came in the example.

So, simulation is started after a QUIT statement or GO statement in the input file. Then post processing is done by the restarting the atlas circuit and device equation simultaneously solved using fully covered algorithm, it is with the Newton's method. So, full Newton meter dot absence full Newton rapid convergence with good initial gas always used for transient simulation. There is a modified method, so there is option M2LN.

So, 2 level Newton method it is less sensitive initial guess and then these are the suffix that are used for indicating the values. So, F for femto, P for Pico, N for nano, U for micro, M for mega, K for kilo and M for mega it is MG, M is for micro milli and MG for mega, G for giga, T for terra.

(Refer Slide Time: 20:18)

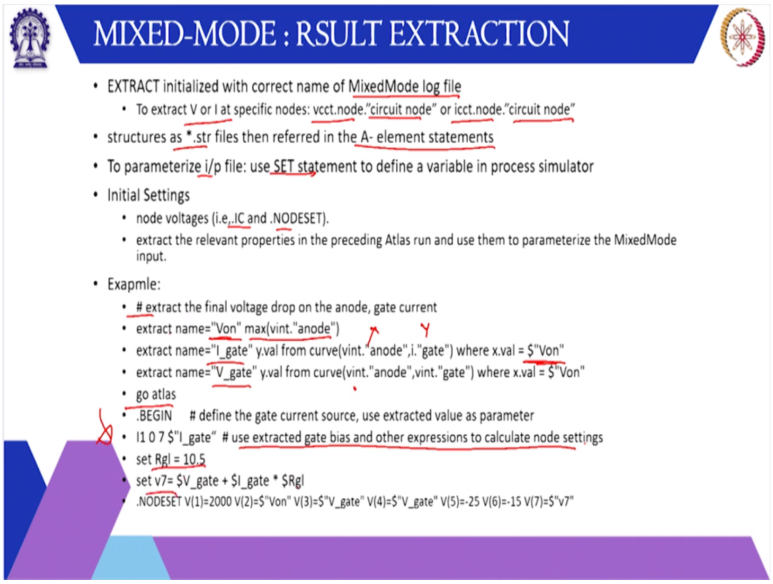


So, multiple Atlas devices such are merged together internally, single output solution file content both the structures. Now first structure referenced is on the top rest are attached below. So, this is a structure nature of the file out for file, so example a diode and bipolar transistor are numerical devices here, so ABJT and ADIO and these are the terminals that are connected one is connected to BASE 2 is node 2 is connected emitter node 4 is connected to collector the width of the device the INFILE.

Similarly, for diode the width INFILE. So, after out putting the solution with dot SAVE MASTER mas the solution file for the first DC point contains both the structure with the second atlas device shifted downwards. So, coordinate shift should be accounted when extracting the position. So, it is one structure file which contain both the structures and the second one is shifted down.

So, if you save the structure file then the parameters have to be carefully X you have to care for while extracting the parameters.

(Refer Slide Time: 21:19)



Then to extract the result initialize with the correct name of the mixed mode log file then to extract voltage of current is specified nodes vcct dot node circuit node or icct node the circuit node. Then structure is saved a dot structure file then reference referred in a statement element statement, then to parameterize the input file you can use SET statement. So, set will basically define a variable and then this variable can be used with dollar name.

So, node voltage are set using dot IC NODESET, you can accelerant parameter, so this is example. So, hash is basically command so hash extract the final voltage drop on the anode or the gate current so the extract name is Von current maximum of V internal dot anode, so at the anode what is the maximum of V internal. You can extract I gate; you can extract V gate so y value from this curve so X versus Y, this is Y versus X and so on.

Where X value is dollar Von, Von is obtained from here. So, now Von is the value the value is accept extracted using this dollar command, then of course we write go atlas dot BEGIN. Then this is the use extracted gate bias other two calculator node setting, so this means we are already using some extracted value to run the simulation. So, this is more like adaptive then you can set Rgl to 10.5 set v7 to dollar V gate dollar I get plus dollar Rgl. Then dot NODEST and so on.

(Refer Slide Time: 22:56)

So, this is basically you know we have briefly discussed about the mixed mode facility in the ATLAS TCAD and with that we you know cross this success this course, thank you very much for your interest. And if you have any queries, we will have you know third session interactive session in the month of April and I look forward to clearing all the doubts that can be cleared. And with this I wish you very best luck best of luck for the exam and thank you very much.