**Basic Electronics**
(Module – 1 Semiconductor Diodes)
Dr. Chitralekha Mahanta
Department of Electronics and Communication Engineering
Indian Institute of Technology, Guwahati

Lecture - 3
Diode Equivalent Circuits

In the last class we have seen the PN junction diode and its characteristic curve. That is with the forward biasing how the current rises almost exponentially and in the reverse voltage when it is applied current is almost zero except for the reverse saturation current due to minority carriers. In order to be applied in a circuit we must get the equivalent circuit for the PN junction diode. That is we have to model the diode electrically. We will be now considering the circuit model for a PN diode. The diode equivalent circuit is a combination of elements properly chosen to best represent the characteristics of a device in a particular operating region.

(Refer Slide Time: 1:57)
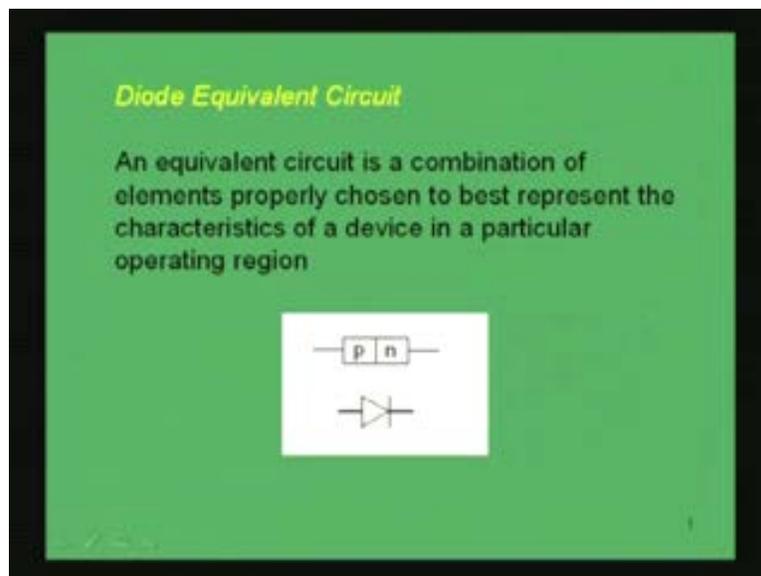
We have this PN junction which is formed from P type and N type materials on silicon or germanium semiconductor and this is the diode symbol that we have seen in the last class. We have to now form an equivalent circuit by a combination of electrical elements. These components must be properly chosen in order to represent the characteristic which is shown by the diode and that should be also in the operating region where we are going to apply the diode.

How to form the equivalent circuit model for a diode? If we consider the characteristic of a diode the forward region we will be seeing this curve which is a non-linear curve. Up to the threshold voltage 0.7 volt due to the barrier potential, current is zero and as soon as

the voltage applied across the diode is beyond or greater than 0.7 volt for silicon, we are taking an example of the silicon, the current rises sharply and this is the characteristic curve and in the reverse bias this is almost zero.

(Refer Slide Time: 3:28)

Piecewise Linear diode characteristic
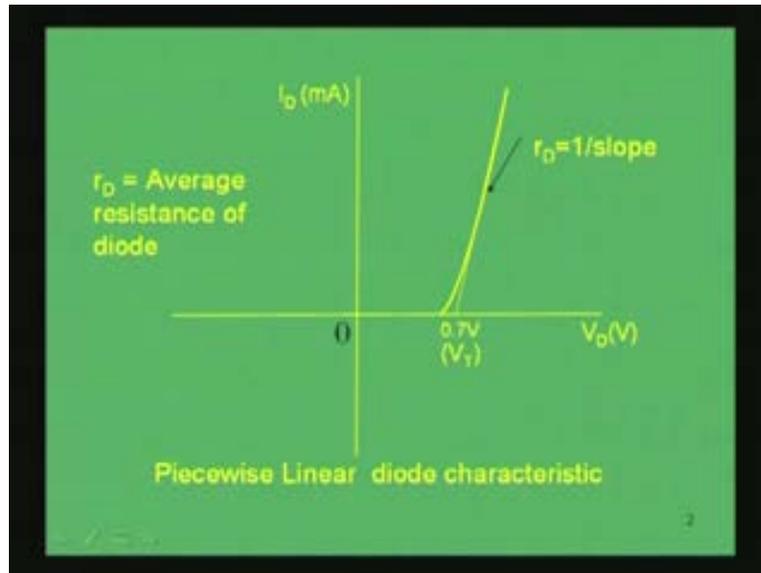
That is why it will be coinciding with the zero line. We do not consider the reverse saturation current. Because it is in the micron range, very small it will not be significant. If we study this characteristic curve of the PN diode we see that this is non-linear and it is not easy to model a non-linear characteristic. Here we have to do an approximation. That is we will be considering this whole characteristic as a Piecewise Linear characteristic. Piecewise Linear characteristic assumption has to be made in order to correctly model the diode. In this case you see that up to 0.7 volt current is zero and then it is rising. We can assume that up to this voltage 0.7 it is zero and then this portion where it is a non-linear characteristic. But when the current rises after sometime you see that it is rising almost linearly. That is why we can approximate this region as a linear characteristic and the overall characteristic can be basically divided into 2 regions. One is up to point 0.7 volt when the current is zero and then a linear characteristic which will be determined by the slope. The average resistance of this diode is the reciprocal of the slope. The slope is actually the y by x and we will be finding out the average resistance by taking its reciprocal because it is voltage by current.

This average resistance of the diode, if we know the two operating point then we can find this out. We will approximate this as the Piecewise Linear characteristic where this average resistance of the diode is to be found out. If you know one point which is say 0.8 volt and 10 milliampere in the operating region that is the point can be known from the data sheet which says about the voltage and current levels for a particular operating point. If we know one point then it is enough to find out this average resistance because r average is del V_D by del I_D that is the rate of change of voltage by rate of change of current between two operating points.

(Refer Slide Time: 6:32)
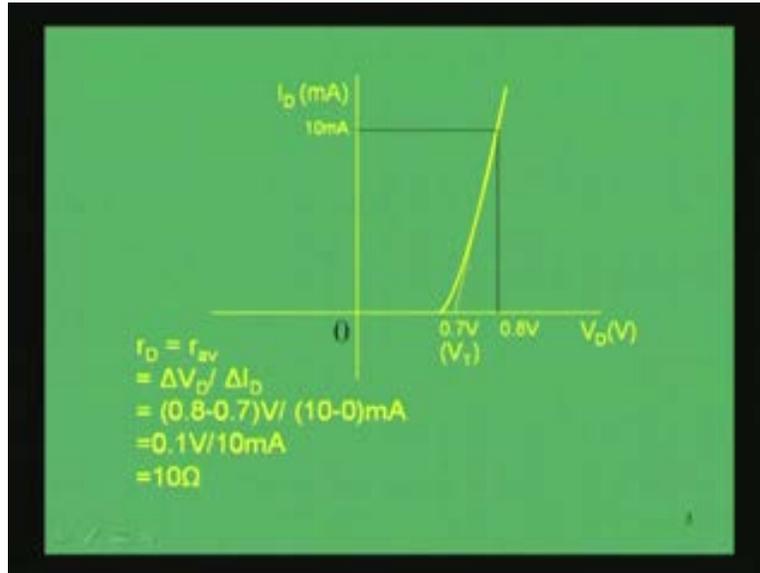
At this operating point if we know the voltage say 0.8 volt and current is 10 milliampere which can be known easily if we see the data sheet of particular diode that we are using, the other point can be taken as this point where the voltage is point 0.7 volt and current is zero. This is a threshold point after which the current rises. As this region is linear that we are approximating by Piecewise Linear approximation we can find from these two points the average resistance. That can be found out if we divide the change of voltage by the change of current between these two points.

One point is this one, one point is this one here the voltage is 0.8 volt and this threshold voltage up to which the current is zero is 0.7 volt. The difference is 0.8-0.7 volt divided by at this point the the current is 10 milliampere and at this point the current is zero. This difference between the currents at the respective points is 10 milliampere minus zero milliampere which is nothing but 10 milliampere. We get this ratio between 0.1 volt and the 10 milliampere that will give us 10 ohm. The average resistance for this particular case we can find out as 10 ohms. This is the average resistance or the order of this resistance of a diode is around this ohm. That is it will be in the order of ohm. Here we have got 10 ohm. The information of this average resistance and this cut in voltage or threshold voltage which is around 0.7 volt for silicon and 0.3 volt for germanium, these two things are enough information to model the diode. That is the cut in voltage and this average resistance.

Piecewise-linear model of the diode forward characteristic is giving us the equivalent circuit representation which is shown here in figure b.

Piecewise-linear model of the diode forward characteristic and its equivalent circuit representation.
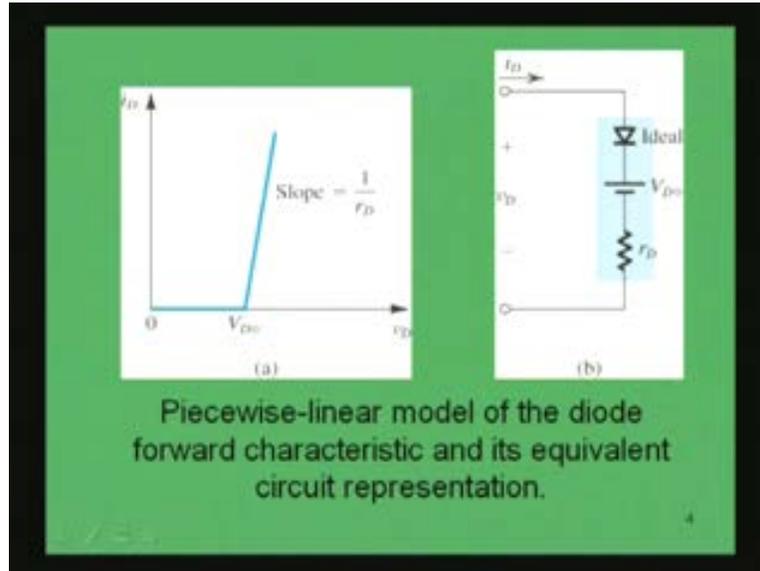
This figure 'a' is the Piecewise Linear model. This V_{DO} means this is the threshold voltage or cut in voltage because of barrier potential. Up to this the current is zero and then this is a piecewise linear approximation that we have done where the slope of this line is 1 by r_D. This is the average resistance of the diode. This straight line is having a slope of 1 by r_D that is enough for finding of the equivalent model. If we draw the equivalent circuit for this Piecewise-Linear model of the diode we show it by two things one is this cut in voltage and one is this average resistance and the direction of conduction in the diode is shown by this ideal diode symbol. This ideal diode symbol is to show the current conduction part and as this barrier potential or this threshold voltage is opposing this conduction of current in this direction this opposition is shown by this battery which is V_{DO}. That is this barrier potential or threshold voltage or cut in voltage the same name denotes this cut in voltage only. Different names are used for this voltage and this is the average resistance. This whole part will be the diode representation and in an electrical circuit we denote it by this model.

This is V_{DO} you know that is 0.7 volt for silicon and this average resistance we have to find out. Given two operating points we can find this out and this ideal diode is just to symbolize this direction of conduction of this diode. This model we have found will be representing the characteristic of the diode in the piecewise-linear approximation. Another approximate model which will be coming from this model again we can further approximatize this. You can further simplify this model which will be giving rise to Constant Voltage Drop Model.
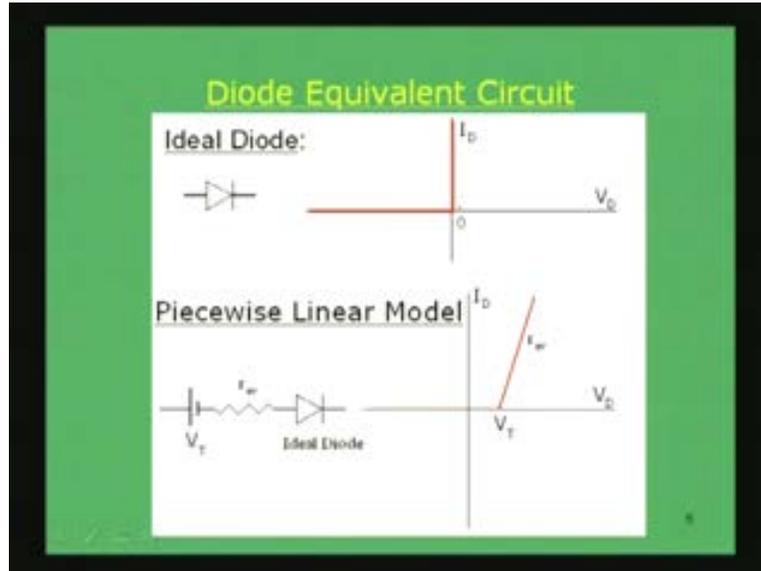
(Refer Slide Time: 11:37)
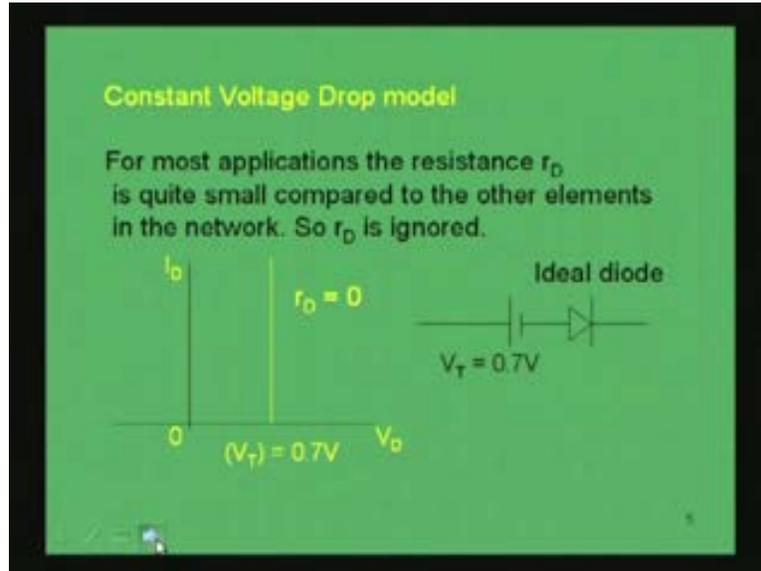
This model is Constant Voltage Drop Model because of the fact that in most of the circuit applications where you generally apply a diode you will find that the resistance of the diode is quite small as compared to the other elements of the network. We can ignore this resistance of the diode without violating much and this assumption will be used for further simplification of this Piecewise-Linear model which gives rise to the constant voltage drop model. That means we do not have this average resistance of the diode into consideration. We do not take that into consideration and we ignore this. That means r_D is ignored and it is made zero. In the characteristic if we now see there will be this 0.7 volt for silicon which is threshold or cut in voltage and the characteristic curve will be rising vertically because this resistance is zero. It will be rising to infinitely high current and this representation we get after simplifying the piecewise-linear model with the assumption that r_D is zero which can be ignored or neglected.

Our new characteristic which will be boiling down from this earlier Piecewise- Linear model is this one. To represent this model we have only to represent it with this cut in voltage which is 0.7 volt for silicon and this ideal diode will be there to denote the flow of the current while the diode is conducting. Here this r_D will be absent. This is the simplified or Constant Voltage Drop Model and this model is very much used in practical circuits and here this resistance is not there. So the model becomes very simpler and in cases where the calculation of current is not so much important and only the application of diode is important that is for example in rectified circuits where we are concerned about the effect of using the diode not the exact calculation of the currents as such then we can use this model and this is mostly used in those cases and that means we are getting 3 equivalent circuits. That is one is the ideal assumption where the characteristic will become like this. That means when the applied voltage is reverse biasing it will be totally zero current and when it just increases beyond zero, that means we applied forward bias the current will rise immediately to infinite value and there will be no voltage drop across the diode.

This is the Ideal Equivalent Circuit for the diode and other model which we have just now found is Piecewise-Linear model. In this case this characteristic will be obtained up to cut in voltage V_T that is a threshold voltage 0.7 volt for silicon, the current will be totally zero and then it will be rising and the rise of this current will be determined by the slope that is the average resistance which is the reciprocal of the slope. This average resistance will determine the rise of this current. The current will rise as per the slope given by this average resistance. This is the Piecewise-Linear model.

This is first model and further simplification from this piecewise-linear model will be the Constant Voltage Drop Model. It is approximate equivalent circuit from the Piecewise-Linear model. In this Constant Voltage Drop Model we have the cut in voltage 0.7 volt and the ideal diode will be denoting the conduction part of the diode.

(Refer Slide Time: 16:16)
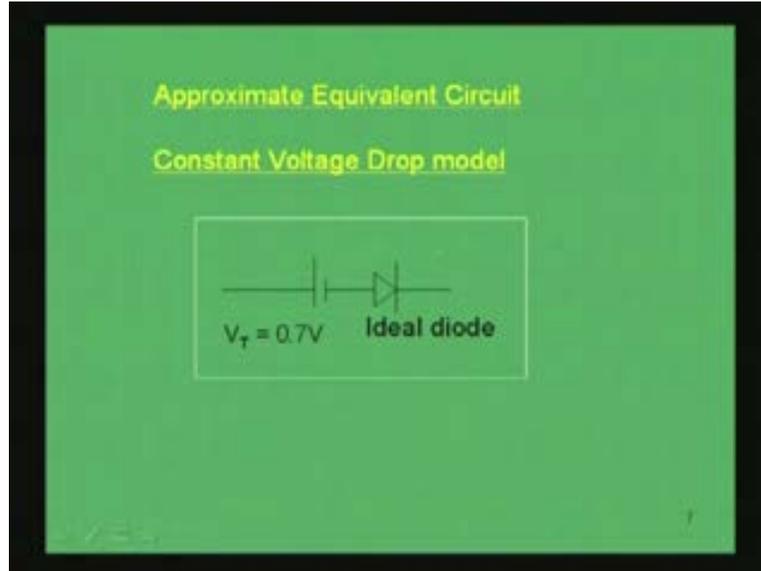
In the Diode Circuit Analysis when you will be applying a diode in an electrical circuit having other components as well the diode will be in the ON state if the current established by the applied sources is such that its direction matches to that of the arrow in the diode symbol and if the voltage applied across the diode is greater than or equal to 0.7 volt for silicon and 0.3 volt for germanium then the diode will be ON.

(Refer Slide Time: 16:55)
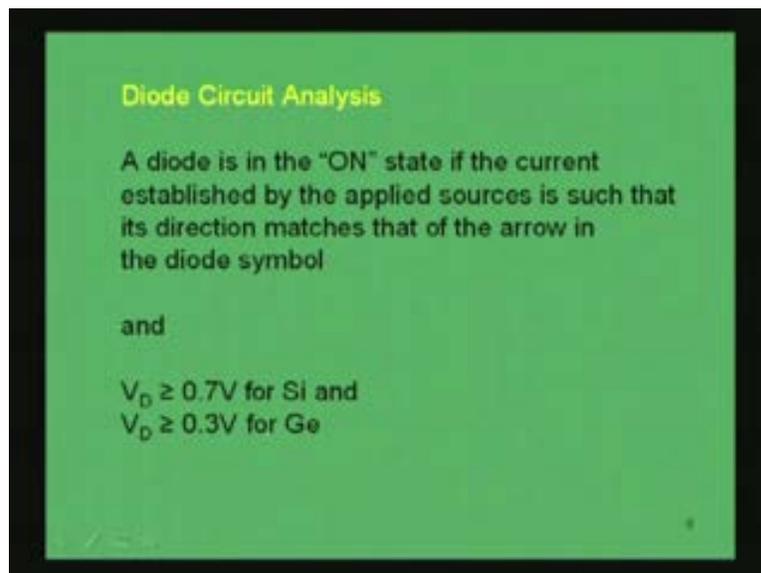
In a circuit when we apply the diode we have to see how much voltage is being applied across the diode and in what direction. Whenever it is beyond 0.7 volt for silicon and 0.3

volt for germanium then the diode will conduct and the direction of current will be given by the direction of the diode current when it is in the ON state that we can find out.

If we now consider a diode circuit, very simple diode circuit let us take having an applied voltage V_{DD} that is a battery and the circuit is having a diode and a resistance R. This resistance is an external resistance.

(Refer Slide Time: 17:50)
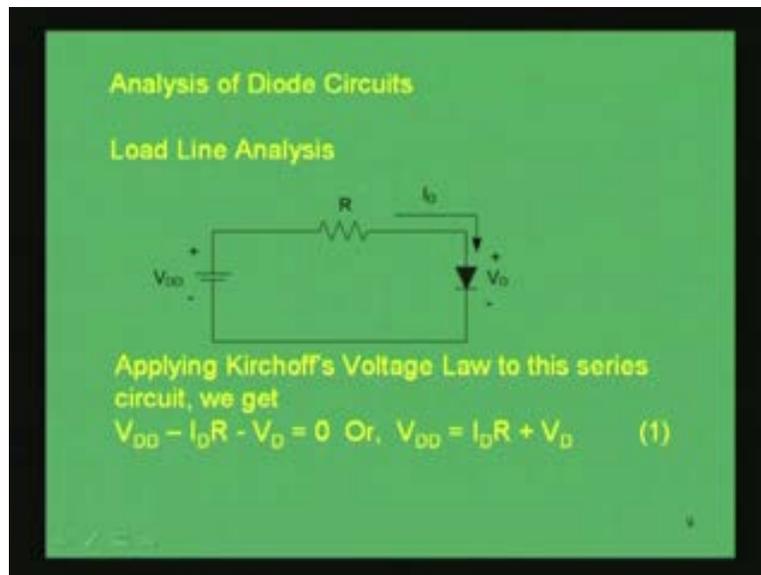
This series circuit if we consider, the approach which is used for analyzing diode circuit graphically is called Load Line Analysis. That means we will have to draw a Load Line. If we consider only resistance as the component which is connected in the diode circuit then the load will be the only resistance and for simplicity we will be considering a resistive circuit. This is a dc circuit. Only component which will matter is resistance and if we consider the series circuit, we will be analyzing this circuit graphically, drawing a load line. How the load line can be drawn? What is the way of drawing this load line that we will discuss first.

In the series circuit we apply the Kirchoff's voltage law which is the law applied to a series circuit and all of us know what Kirchhoff's voltage law is. The algebraic summation of the voltage drop in this loop is equal to zero. That law if we apply in the circuit we will get V_{DD} minus I_D into R. I_D is the current in this circuit, diode current and minus V_D the drop across this diode is equal to zero. It is coming from the Kirchoff's voltage law applied here in this circuit. Simplifying this we get V_{DD} equal to $I_D R$ plus V_D. In this equation if we now set V_D is equal to zero, we get the maximum diode current that can flow in the circuit given by I_D equal to V_{DD} by R. That will be one extreme point that denotes the maximum diode current that can flow in the circuit and if we make I_D equal to zero in equation one, this equation if we make I_D equal to zero then we get the maximum voltage across this diode. That is V_D equal to V_{DD}.

(Refer Slide Time: 20:26)

If we set $V_D = 0$ in Eq.(1), we have
$$V_{DD} = I_D R$$
and $I_D = V_{DD}/R$

If we set $I_D = 0$ in Eq.(1), we get
$$V_D = V_{DD}$$

These two points will define a straight line and this is called the load line

The intersection of the load line with the diode characteristics will determine the point of operation of the system

These two points will be defining a straight line because in order to draw a straight line we need only two points and this straight line is called the load line and if we now draw the characteristic curve and superimpose on it the load line then there will be a point of intersection between these two graphs and that intersecting point will be the point of operation of the system. That means we can find the point of intersection and that point will denote the point of operation for the circuit. Let us take an example and see. Before that this is the diode characteristic.

(Refer Slide Time: 21:09)
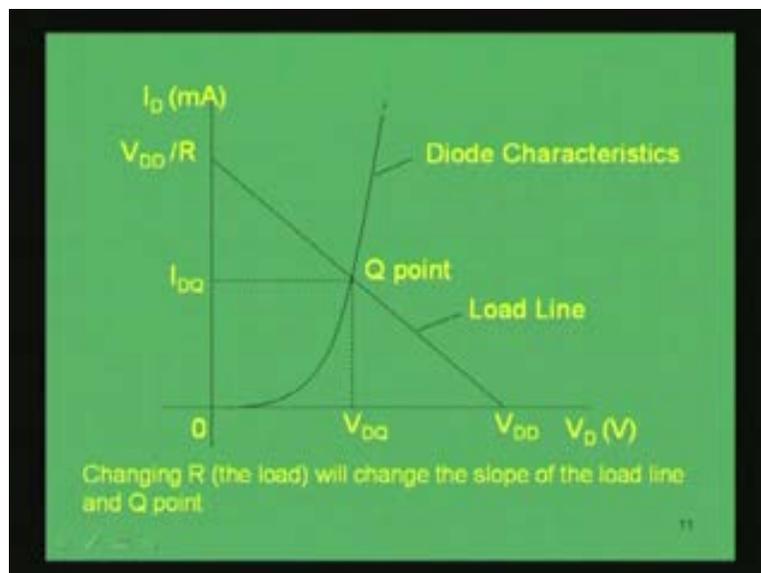
This plot will be the diode characteristic, non-linear characteristic. We draw the load line in the same graph. These two points, one is V_{DD}, one extreme point and the other is V_{DD} by R, the other extreme point for this load line. Connect these two points and we get the straight line. This straight line and the characteristic curve are intersecting at this point which is the Q point. It is called the Q or quiescent point or it is silent point. It is also called the steel point meaning that only a dc voltage is applied. If a small signal voltage is applied along with the dc then the Q point will shift up or down but here will be the only the shift condition and that will be called the silent point or Q point.

This is the operating point for your circuit. From this Q point, if I project on the x- and y-axis the voltage and current axis, I get these two points V_{DQ} and I_{DQ} and these two points will determine exactly the Q point. What is the Q point? That point, is determined by this voltage V_{DQ} and the current I_{DQ}. I know now the operating point having this voltage across this diode and the current through this circuit of the diode is I_{DQ}. If we change this resistance in the circuit then slope of this load line will change. We will be getting a different load line giving a different Q point. Consider a circuit, a series diode circuit given in figure 1.

(Refer Slide Time: 23:05)

Fig.1(a)

Fig.1(b)
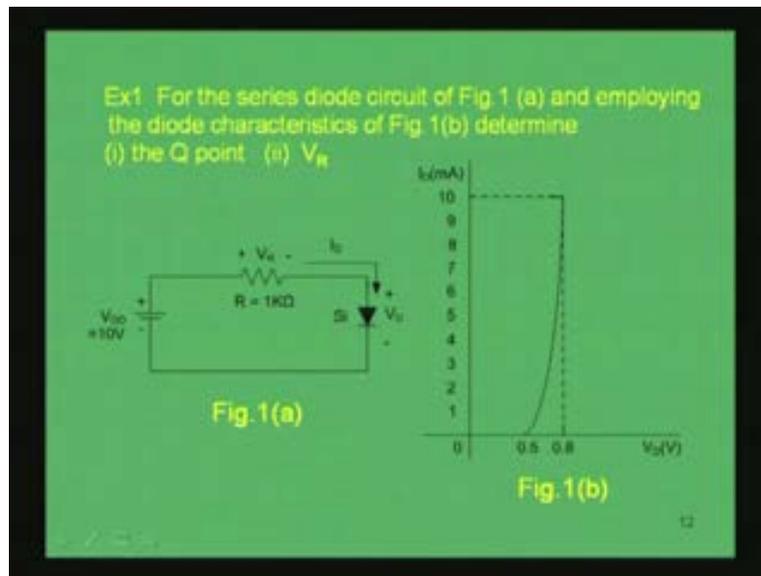
Here the applied voltage V_{DD} is 10 volt, resistance applied externally in the circuit is 1 kilo ohm and this is the diode which is a silicon diode; it is mentioned here. This diode which is used in this circuit is having this characteristic given by the figure 1(b). This is the characteristic curve of the diode. You can see that after 0.5 volt, the current starts to rise and at 0.8 volt it has 10 milliampere current flowing through it. This characteristic curve is given to you. For this circuit employing this diode characteristic you have to determine the Q point as well as the voltage across this resistance V_R. These two are to be found out. For that first of all you have to find out the two extreme points of the load line. We have already seen the extreme current I_{Dmax} will be given by V_{DD} by R when you

make this voltage drop across the diode zero and for this particular example it is 10 volt by 1 kilo ohm.
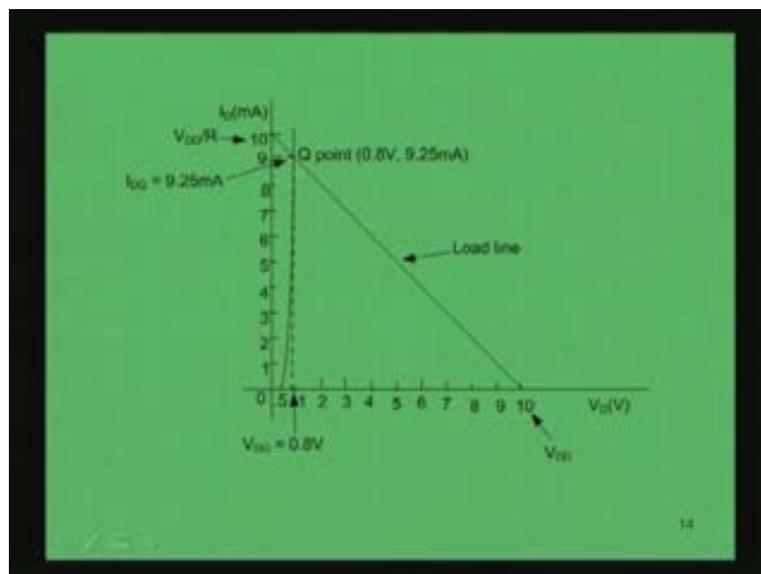
(Refer Slide Time: 24:37)

So it is 10 milliampere and V_{Dmax}, this voltage can be found out making this I_D zero that is given by the applied voltage V_{DD} so this is 10 volt. These two points are known. We can now draw the load line. Let us do that. This load line is drawn.

(Refer Slide Time: 24:59)

One point is 10 volt V_{DD} max, maximum V_D and this point is I_{Dmax} that is V_{DD} by R which is 10 milliampere. Connecting these two points I get a straight line. This is the load

line which is drawn along with this characteristic curve. As you can see here the point of intersection between this characteristic curve, this load line is Q. If I now project vertically from this Q point, I get the point of intersection with the V_D axis at around 0.8 volt. This whole plot has to be drawn to scale only then the exact point of intersection can be found out. Otherwise there will be an error. If I project it on the other axis, current axis here it intersects that at around 9.25 milliampere. This Q point is defined by 0.8 volt and 9.25 milliampere. This is the operating point. That means this circuit current at that particular operating point is 9.25 milliampere and the voltage across this diode is 0.8 volt. What about the voltage V_R that has to be found out? The voltage V_R as you can see in this figure, this voltage drop is the current flowing through this resistance multiplied by this resistance. Current flowing through it is I_{DQ} at that particular operating point and multiplied by resistance 1 kilo ohm we get this. V_R that is equal to 9.25 milliampere into resistance 1 kilo ohm which is 9.25 volt. This is one way of finding out V_R. Another way also we can find out this V_R. This V_R is nothing but this voltage minus this drop. There is another way. Both the ways would actually match. If we correctly measure this graph, this would be same. V_R is equal to V_{DD} minus V_{DQ}. 10 volt minus 8 volt is 9.2 volts. There is a difference. But ideally if we measure the graph correctly both will be same. This error comes because there is an inaccuracy in reading the graph. When you measure it to scale it should be same. Both the results should be same ideally. That should be same in both the cases.

(Refer Slide Time: 27:49)

In this whole analysis we have seen that it is very important to draw the characteristic curve and measure everything accurately. That is tedious or in some cases you do not really need so much of accurate calculations. We can also use an approximate equivalent model for a diode. If we consider again this characteristic curve which is non-linear, the exact characteristic curve is this one. But this can be approximated by the constant voltage model. Constant voltage model we have seen that 0.7 volt is the cut in voltage. If I use that model instead of this whole non-linear characteristic curve which is very

difficult to accurately measure and it is not needed also. We draw this approximate equivalent characteristic that is 0.7 volt and current is rising infinitely at this point. After 0.7 volt is overcome it is rising infinitely. That is the approximate equivalent model. Then let us see where this point of intersection is lying?

(Refer Slide Time: 29:07)
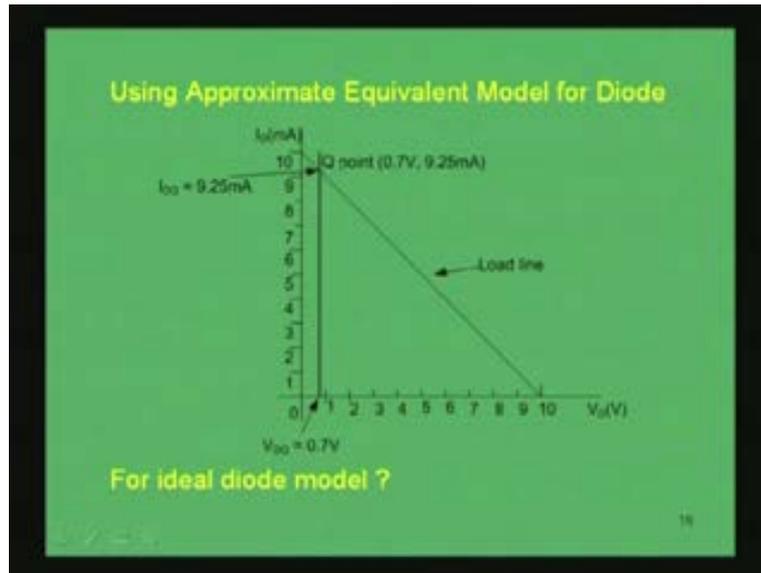
Here we can see that Q point is defined by 0.7 volt and this current is also found to be 9.25 milliampere. This is the upper Q point we have got using Approximate Equivalent Model. Using Approximate Equivalent Model whatever Q point we have got that is not very much different from using exact equivalent circuit because here earlier we got this voltage as 0.8 volt but here we are getting 0.7 volt but the current is same. In this case we can see very clearly that using approximate equivalent model will also serve our purpose and where this exact calculations of current is not required or exact voltage calculation is also not required we can use this Approximate Equivalent Model and carry on our analysis. You will find that in most applications this Approximate Equivalent Model or Constant Voltage Model is used.

What will happen if you use ideal model? In ideal diode model current will be raising at zero point itself and the point of intersection will be at 10 volt. This is quite different than the results of approximate equivalent circuit because if I consider this line and find out this intersecting point, the intersecting point is at 10 milliampere. 10 milliampere intersecting point in a current point we get and voltage is zero whereas in this case we have seen 0.7 volt and 9.25 voltage. This is very much different from what we will get in ideal model. So ideal model is practically not very much useable and that also we will be using only when we have to know the current conduction part. In that case we can use an ideal model. Effectively we will be using mostly the Approximate Equivalent Model. But what we are considering is about the dc signal, dc constant voltage. But what will happen if we have a small ac signal superimposed on the dc voltage.

(Refer Slide Time: 31:35)


Small Signal model of diode
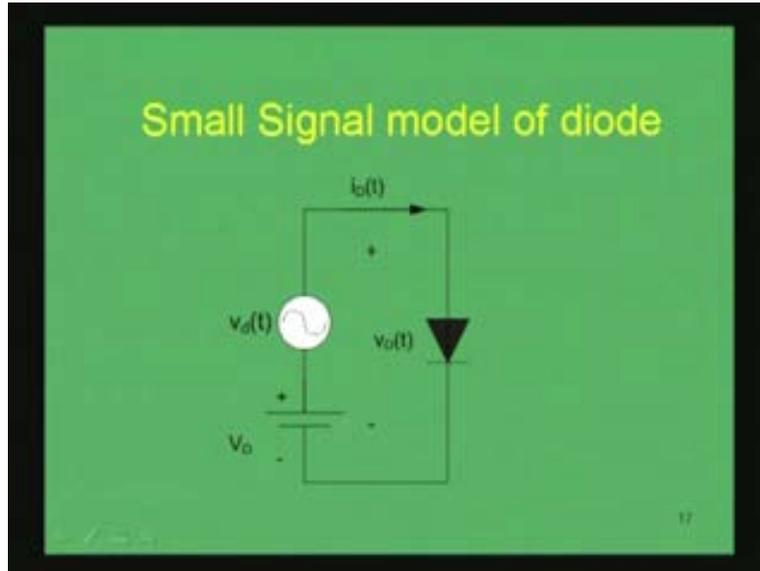
Then our whole model will have some extra terms not only the resistance because whenever this change occurs in the voltage then capacitance effects will come. There are two capacitance effects which will be present in a diode. Those are known as the junction capacitors and the diffusion capacitors. Junction capacitance basically is related to the space charge region and the capacitance will be developed in the space charge region and the voltage across this space charge region or depletion region changes. Let us consider again the PN junction. Go back to PN junction and what will be the depletion layer width?

(Refer Slide Time: 32:27)

Depletion layer width in open circuited P N Junction

$$W_{dep} = \sqrt{\frac{2\epsilon_s}{q}\left(\frac{1}{N_A} + \frac{1}{N_D}\right)V_{bi}}$$
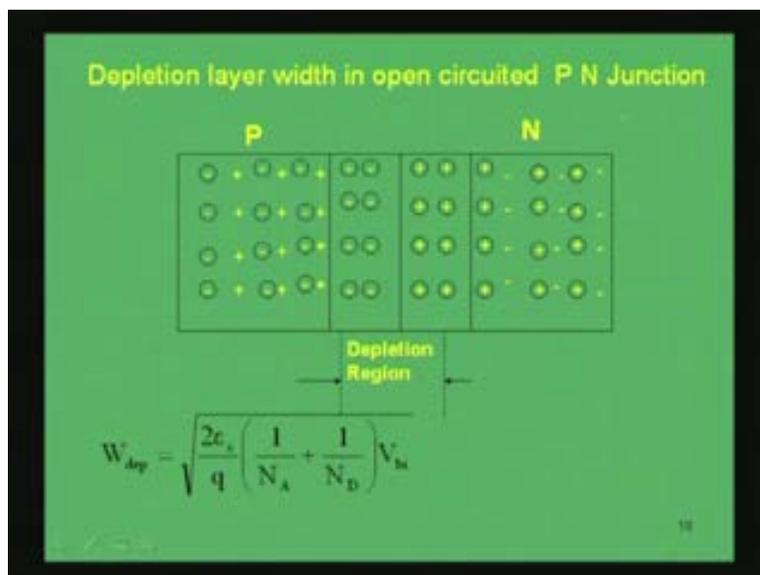
We already know in an open circuited PN junction this will be the depletion layer width given by this expression which we have earlier discussed. If we have forward biased this PN junction, these majority charge carriers they will be repelled. These holes will be repelled by this positive terminal and electrons will be repelled by this negative terminal. They will come towards the junction and in the process they will be neutralizing these ions as a result of which the depletion layer width will reduce.

(Refer Slide Time: 33:05)
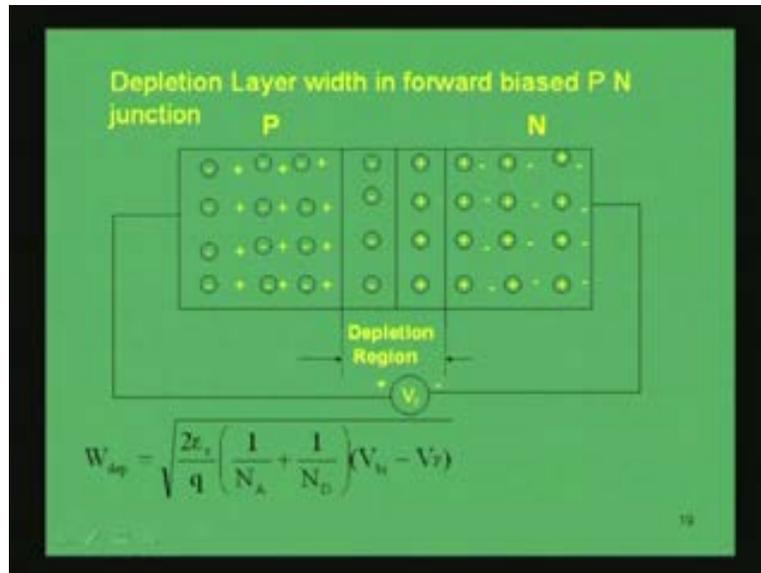
That means there will be less uncovering of the fixed charges. The depletion layer width reduces. The depletion layer width will be given by in the earlier expression of open circuit instead of V_{bi} we will be having V_{bi} minus V_F because the potential barrier is now reduced. The potential hill is reduced by this voltage. The depletion layer width is reduced and if we consider the reverse biased case the depletion layer width will be more since more and more charge carriers will be attracted by this positive and negative terminal of the reverse bias. Electrons will be attracted away from the junction holes will be attracted by this negative terminal away from the junction. Uncovered charges of the ions will be more and more. The charges will be more here, uncovered charges and the depletion layer width also will be affected because the voltage across this will not be V_{bi} but V_{bi} plus V_R. This potential hill is now increased.

(Refer Slide Time: 34:19)
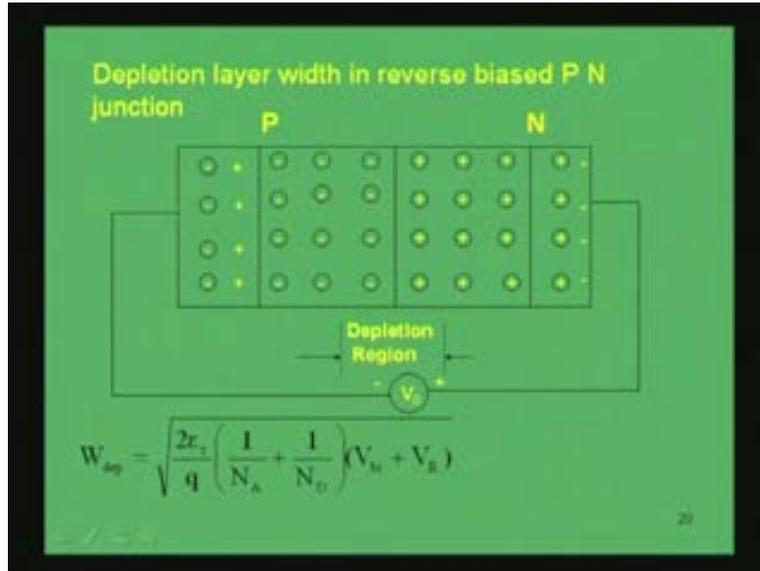
If you visualize this width there are charges on the right positive there are charges on the left negative which are fixed charges and the number of fixed charges are changed by this forward and reverse voltage because forward voltage will be reducing the number of charges accumulated under depletion layer and the reverse bias will cause more charge. That means it is like charging and discharging taking place across a capacitor. If we consider as a parallel plate capacitor, we can visualize that there is a parallel plate capacitor, the width between these two plates are W depletion and we have this in between material silicon and so it is like a parallel plate capacitor, C which is given by E_s A by W_{dep}.

(Refer Slide Time: 35:32)

E_s is the permittivity of silicon, A is the cross sectional area of the junction which is perpendicular to this plane of the paper and this W_{dep} is the depletion layer width. I already know W_{dep}. I am taking this case specifically for reverse biased PN junction because this junction capacitance will be dominant for reverse bias PN junction not for the forward bias PN junction. After replacement the whole expression comes like this and now in the numerator if we consider this whole term I can replace it by C_{j0} means the capacitance when applied potentially zero, open circuit. The whole term will be C_{j0} by root over 1 plus V_R by V_{bi}.

(Refer Slide Time: 36:42)

This expression is just found out after simplification of this and replacing the numerator part by C_{j0}. We get junction capacitors given by this. Here V_R is the reverse voltage. This is the open circuit built in potential. This is called junction capacitance or transition capacitance also. This is very prominent in the case of reverse biasing PN diode and there is another diffusion capacitance which is prominent for forward bias PN junction and this diffusion capacitance C_d is the rate of change of injected charge with voltage and if we consider the small signal model of a diode apart from the resistance, average resistance, there will be two other factors. One is junction capacitance, one is the diffusion capacitance.
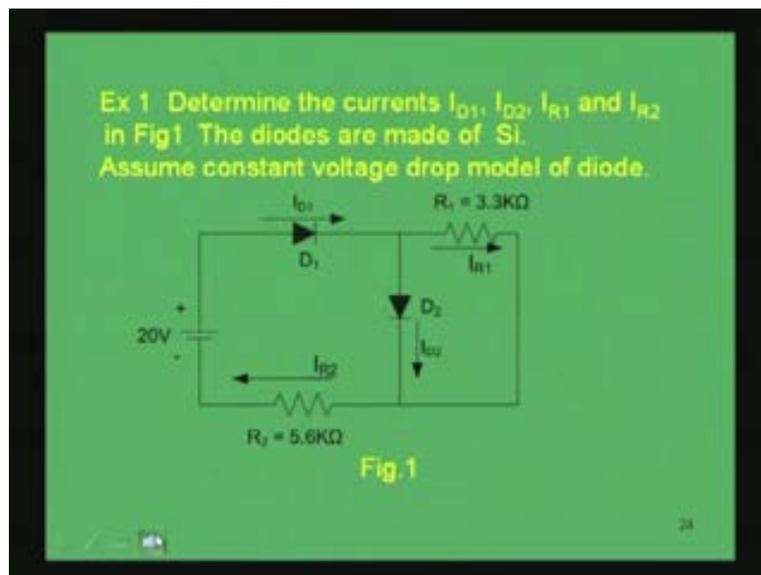
The overall circuit representation for small signal model will be this resistance r_d, small r_d to denote that it is small signal resistance and C_j C_d these two are the capacitances, junction capacitance and diffusion capacitance.

(Refer Slide Time: 38:02)

Small Signal model of diode
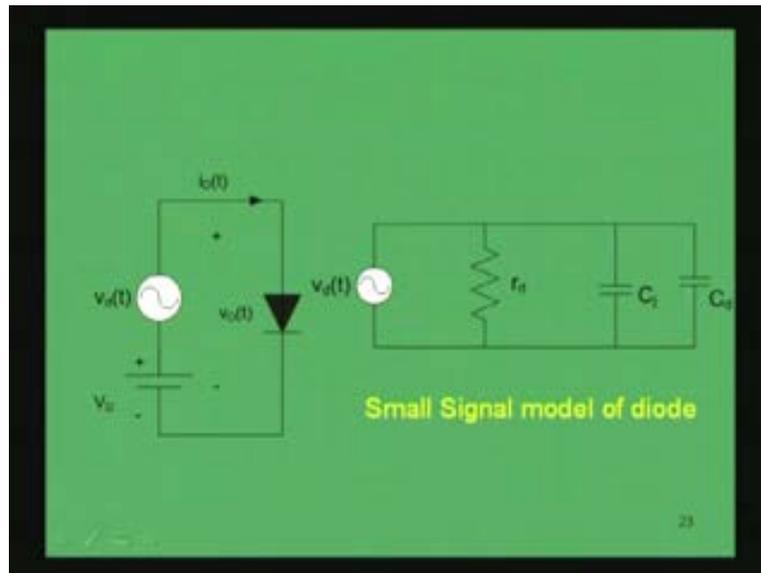
We have this small signal model. When you have a small signal along with the dc voltage that I am applying then we have to consider this model. But for the time being, we will concentrate more on the dc voltage applied only. Let us do an example.

(Refer Slide Time: 38:24)

Ex 1 Determine the currents I_{D1}, I_{D2}, I_{R1} and I_{R2} in Fig1 The diodes are made of Si. Assume constant voltage drop model of diode.

$R_1 = 3.3K\Omega$

$R_2 = 5.6K\Omega$

Fig.1

You have a circuit given by this. There are two resistances, two diodes. You have to find out this current flowing through this diode D_1, current flowing through this diode D_2 as well as current flowing through this resistance IR_1 and current flowing through this resistance R_2. The voltage applied in the circuit is 20 volt and it is given that the diodes

are made of silicon. This information is important for finding out the cut in voltage and you are told that you have to assume constant voltage drop model of the diode. Constant voltage drop model will have this cut in voltage. As it is silicon it will be having 0.7 volt cut in voltage and there will be no resistance considered in the diode. Since the constant voltage drop model does not have the resistance in its model that means it is ignored. Let us solve this example.

We again draw the circuit. Here this voltage is given as 20 volt and this one is diode D_1 and this is D_2. R_1 is given as 3.3 k ohm. R_2 is given as 5.6 kilo ohm. What will be the currents I_{D1}, I_{D2}, I_{R1} and I_{R2}? This is the problem. You have to use constant voltage drop model. First thing is to find out whether the diodes will be conducting or not. As you can see this 20 volt is applied which will be forward biasing the diodes and so this diode will be conducting. When the diode conducts 0.7 drop will be there but 19.3 volt will be available across this. This diode will also be conducting. Both these diodes are conducting. That means we have now the current flowing in this diode also, in this diode also. We have to find out what will be current? The drop across this D_1 which is V_{D1} the voltage drop across D_1 diode and it is a constant voltage drop model so 0.7 volt. Similarly V_{D2} will be also 0.7 volt. These two diodes are made of silicon. So they have same drop across them.

If we consider the voltage drop across this resistance R_1 the voltage drop between these two points is 0.7 volt. If we know this volt divided by this resistance 3.3 k, I can find the current I_{R1}. I_{R1} is equal to V_{D2} by R_1 and V_{D2} is 0.7 volt divided by 3.3 k ohm. This will give you 0.212 milliampere. One current we have found out that is I_{R1}. In this circuit this part this Kirchhoff's Voltage Law is applied. If you apply Kirchoff's voltage law in this loop that is 20 volt minus V_{D1} that is the drop across this diode minus V_{D2} minus V_{R2} where V_{R2} is this drop.

(Refer Slide Time: 43:07)
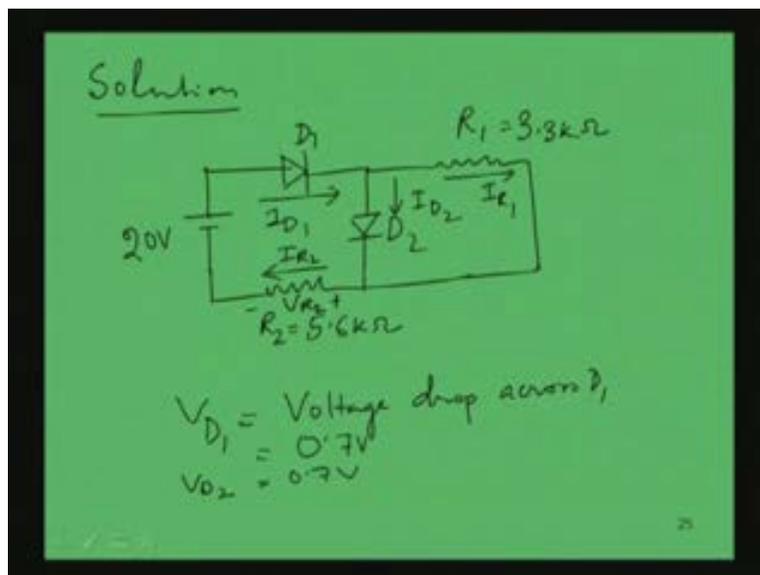
Use KVL in this loop. Then using KVL we get 20 minus V_{D1} minus V_{D2} minus V_{R2} that is equal to zero; everybody knows what Kirchoff's voltage law is. Algebraic sum of the voltage around the loop is zero. We are just simply applying that. You substitute V_{D1}, V_{D2} that you know. 20 minus 0.7 minus 0.7 minus V_{R2} is equal to zero. We can find out what is V_{R2}? 20-0.7-0.7; that is equal to 20-1.4 and so 18.6 ohms is available across this resistance R_2. If I know this voltage I can find this current I_{R2}. What is this current? V_{R2} divided by R_2. I_{R2} is equal to V_{R2} divided by R_2. Just substitute these values. 18.6 volt divided by R_2 is given as 5.6 k. That will be 3.32 milliampere. This is I_{R2}.

(Refer Slide Time: 44:34)

What is left to find out? Let's see. I_{D1} is to be found out. I_{D2} also we have to find out. We have found out I_{R1} and I_{R2}. This current I_{D1} is same current I_{R2} because you have seen here the current which is leaving this positive terminal of this battery must enter this positive and negative terminal of this battery. The current which is coming out from this terminal is I_{D1}. The current which is entering this terminal is I_{R2}. This must be equal. So your I_{D1} is nothing but I_{R2}. I_{R2} you have already found out and that is equal to 3.32 milliampere. What is left is I_{D2}. This I_{D2} has to be found out. Come to this point and you apply Kirchoff's current law to this point.

Kirchoff's current law says that the sum of the incoming and outgoing currents at a node is equal to zero. What are the incoming currents and outgoing currents at this node? Incoming current is I_{D2} that we have to find out and another incoming current is I_{R1}. This is I_{R1} and outgoing current is I_{R2}. So this algebraic sum must be equal to zero. We follow this rule of entering current and leaving current are having opposite signs this leaving current is say a positive I_{R2}. Entering current I_{R1} and I_{D2} are negative. Then we get I_{R2} minus I_{R1} minus I_{D2} equal to zero. What is I_{D2}? That is the question. We have to find out I_{D2}. That is nothing but I_{R2} minus I_{R1}. I_{R2} we already know is 3.32 milliampere minus I_{R1}

is 0.212 milliampere. These two we have found out. This subtraction will lead to 3.108 milliampere. So all the currents are now known.

(Refer Slide Time: 47:11)

In this example we have used constant voltage model. Let us do another example. Try to solve this problem.

(Refer Slide Time: 47:24)
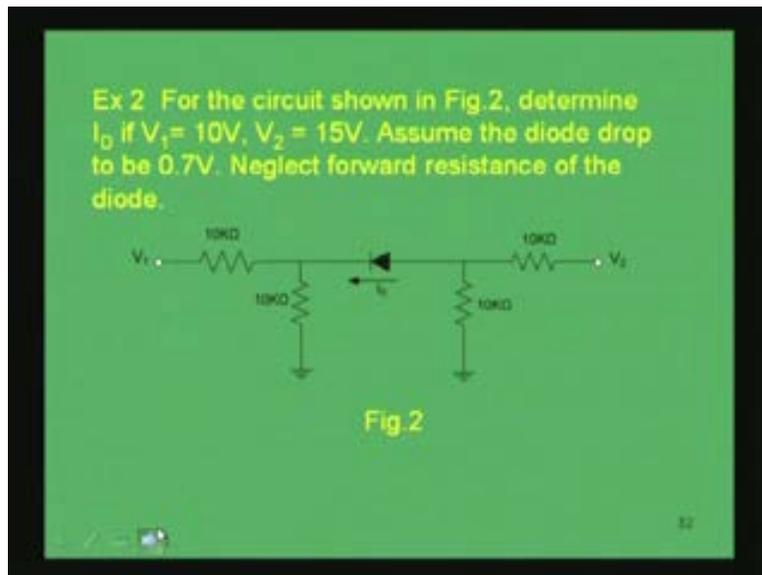
In this circuit shown in figure 2 determine the current I_D, the current which flows through this diode if the voltage V_1 is given as 10 volt, V_2 is given as 15 volt and assume the diode drop to be 0.7 volts. That means here also you have used constant voltage drop model because you have to neglect forward resistance of the diode. In this circuit we have

to find out I_D. We have to first of all examine what is the voltage being applied across the two points of the diode. This is P this is N. What voltage will be applied across these two points? Let us try to solve this problem. Again we draw. This is the diode and the resistances are like this. This is ground, this is ground; this is V_2, this is V_1. The resistances are 10 k, 10 k. All are 10 kilo ohms and we have to find out this diode. This V_1 is equal to 10 volt. V_2 is equal to 15 volt. What voltage is being applied across this diode? V_D is the voltage across this diode. If I consider this voltage V_D what will the voltage?

At this side you have these voltages 15 volt and this side it is 10 volts. The potential across these two points is 15 minus 10 which is 5 volts. V_D is 5 volts and as it is greater than 0.7 volts the diode will conduct. This diode is conducting and that has been ascertained first. We have to find out this current I_D. To find out this current we will be taking help of nodal analyses. Let us consider this node and let us name this node as 'a', this node as 'b'. The voltage at this node with respect to ground is V_a, the voltage at this node with respect to the ground is V_b and the Kirchoff's current law will be applied at these two nodes. Let us consider the node 'b' and find out according to Kirchoff's current law. To the node 'b' apply KCL, Kirchoff's current law. Kirchoff's current law is sum of the incoming and outgoing current is equal to zero. The outgoing current is what; V_b minus V_2 by 10k. Assuming that this outgoing current is in this direction V_b minus V_2 by 10k plus V_b minus zero; this is ground divided by 10k. Another outgoing current is I_D. Summation of all this is equal to zero. All are in the same direction which is outgoing. Applying Kirchoff's law at node 'b' V_b minus V_2 by 10k; I am removing k. All are in kilo ohms order. So I am omitting this kilo plus V_b by 10 plus I_D equal to zero. This is equation number 1.

Similarly if I apply the Kirchoff's current law at node 'a' what will I get? The sum of incoming and outgoing current is equal to zero. Incoming current is I_D and let us take the outgoing current as V_a by 10k plus V_a minus V_1. This is the drop across these two points divided by 10k will give me this current.

(Refer Slide Time: 51:33)
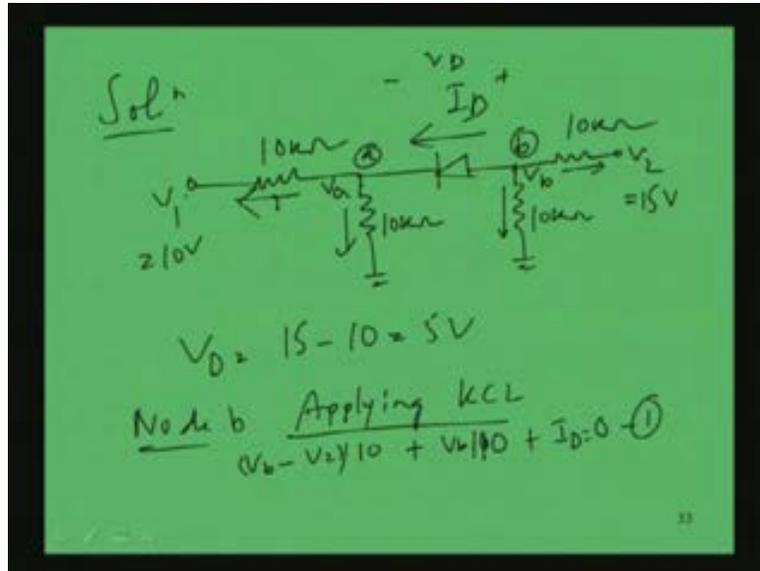
In this second case at node a if i apply KCL, I get V_a by 10 one outgoing plus V_a minus V_1 by 10. If I consider the diode current I_D as it is incoming it will minus I_D that is equal to zero. This is equation number 2. We have to solve number 1 and number 2. I write equation number 1 again. What was the equation? V_b minus V_2 by 10 plus V_b by 10 plus I_D equal to zero. I can sum them up and eliminate the term I_D in solving of this equation. This is equation number 1. Summing up 1 and 2, I get V_b minus V_2 by 10 plus V_b by 10 plus V_a by 10. This and this will cancel plus V_a minus V_1 by 10 is equal to zero. This V_2 and V_1 we know. What is V_a; V_b minus this drop. This drop is 0.7. So V_b minus 0.7, the diode drop that is equal to V_a the difference in the potential. V_b minus this drop is equal to this potential. We can now replace this V_a in terms of V_b. V_b minus 0.7. Constant voltage drop model we are using and it is given that V_1 is equal to 10 volt, V_2 is equal to 15 volt. If I substitute in this equation now I can solve it. From equation 3 I will be having V_b minus V_2 is equal to 15.

(Refer Slide Time: 53:47)

Node a   KCL

$$\frac{V_a}{10} + \frac{V_a - V_1}{10} - I_D = 0 \quad -(2)$$

$$\frac{V_b - V_2}{10} + \frac{V_b}{10} + I_D = 0 \quad -(1)$$

$$(1) + (2) \Rightarrow \quad \frac{V_b - V_2}{10} + \frac{V_b}{10} + \frac{V_a}{10} + \frac{V_a - V_1}{10} = 0 \quad -(3)$$

$$V_a = V_b - 0.7$$

$$V_1 = 10 v \qquad V_2 = 15 V$$

V_b minus 15 by 10 plus V_b by 10; keep V_b as V_b. I will substitute V_a; plus V_a is V_b minus 0.7. Here I substitute. I want to keep only one unknown plus V_a is V_b minus 0.7. V_b minus 0.7 minus 10; V_1 is 10 divided by 10 equal to zero. Here only one unknown is there which is V_b. We can very easily solve this. It will be further simplified V_b by 10 minus 1.5 plus V_b by 10 plus V_b by 10 minus 0.07 plus V_b by 10; that is 10.7 by 10 is 1.07; that is equal to zero. I get 1, 2, 3, 4V_b by 10 and the other side if we do this calculation I will be getting 2.64. That gives me V_b equal to 2.64 into 10 is 26.4 divided by 4 that is giving me 6.6 volt. Now I get V_b equal to 6.6 volt.

(Refer Slide Time: 55:22)

$$\frac{V_b' - 15}{10} + \frac{V_b}{10} + \frac{V_b - 0.7}{10}$$
$$+ \frac{V_b - 0.7 - 10}{10} = 0$$

$$\text{or,} \quad \frac{V_b}{10} - 1.5 + \frac{V_b}{10} + \frac{V_b}{10} - 0.07$$
$$+ \frac{V_b}{10} - 1.07 = 0$$

$$\rightarrow \quad \frac{4V_b}{10} = 2.64 \Rightarrow V_b = \frac{26.4}{4}$$
$$\underline{V_b = 6.6 V}$$

If I know V_b, I can find out what is I_D? What is this I_D? Here I can substitute V_b which I have found out. So what will be I_D? Transfer all this left side to the right. It will be V_2 minus V_b by 10 minus V_b by 10. I_D is equal to V_2 minus V_b by 10 minus V_b by 10. Replace these values V_2 minus V_b by 10. V_2 we have already given in the circuit as 15 volt. 15 minus V_b we have found out to be 6.6 volt; divided by 10 minus again V_b by 10 is 6.6 by 10. That is 10 in the denominator and in the numerator 15 minus 6.6 minus 6.6. Do this calculation. It will give you the numerator as 15 minus 13.2 by 10. So you will be getting 1.8 volt in the numerator. In the denominator remember the 10 is 10 kilo ohms actually. I have omitted the kilo in the earlier stages; voltage by kilo will give the unit in milliamps. What will be the result? 0.18 milliamps. I_D, direct current which I have got is 0.18 milliampere.

(Refer Slide Time: 57:18)

$$I_D = \frac{V_2 - V_6}{10} - \frac{V_6}{10}$$

$$= \frac{15 - 6 \cdot 6}{10} - \frac{6 \cdot 6}{10}$$

$$= \frac{15 - 6 \cdot 6 - 6 \cdot 6}{10}$$

$$= \frac{15 - 13 \cdot 2}{10} = \frac{1 \cdot 8 V}{10 k\Omega}$$

$$I_D = 0 \cdot 18 mA$$

In this example we have taken help of only the primary laws or principles like Kirchoff's voltage law, Kirchoff's current law. Kirchoff's voltage only we have been using. Whenever you analyze the circuit all those basic primary laws, fundamental laws of electrical engineering like Kirchoff's current law, voltage law, nodal analysis all these will be applied and the diode should be replaced by its model to fit into the circuit. It is not a physical diode. We will consider the approximate equivalent model of the circuit that is the electrical circuit which will be representing the diode.

Today we have learnt about how modeling is done on the physical diode, PN junction diode. Approximate equivalent model of the diode has been derived which is most commonly constant voltage model that is being used in electrical analysis of this circuits having diode. When you have to only consider the diode in application only that is we are not very much concerned how accurately we find out the current and voltages that is if it is not a prime concern we will use approximate model and even if approximate then the constant voltage model is the ideal model of the diode which is very basic and it will be only used for those cases of circuits when the applied voltage is much, much more or the other components of the circuit is much higher in values than the resistance of the diode then we will be ignoring the diode resistance. We will only consider the ideal model of the diode without considering the cutting voltage as well as the resistance. We will be using it only as a switch like it is either conducting or non-conducting. In that respect we will use ideal diode but generally we use constant voltage model. We will use this model, constant voltage model as well as the ideal model as well also the Piecewise linear model of the diode. What is the average voltage, average resistance if it is mentioned then we will use the Piecewise linear model if the compulsion is there that we have to use the average resistance also. Otherwise we will be sticking to the constant voltage drop model of the diode and next we will be going into the detailed discussion on application of diode in the circuits like rectifier circuits, etc.