Water Resources Engineering

Professor P.K. Mohapatra

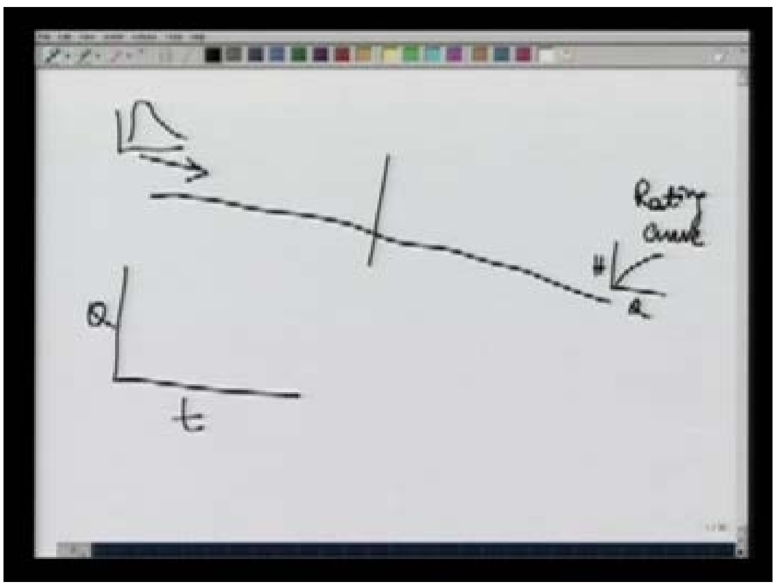
Department of Civil Engineering

Indian Institute of Technology, Kanpur

Lecture No. # 29

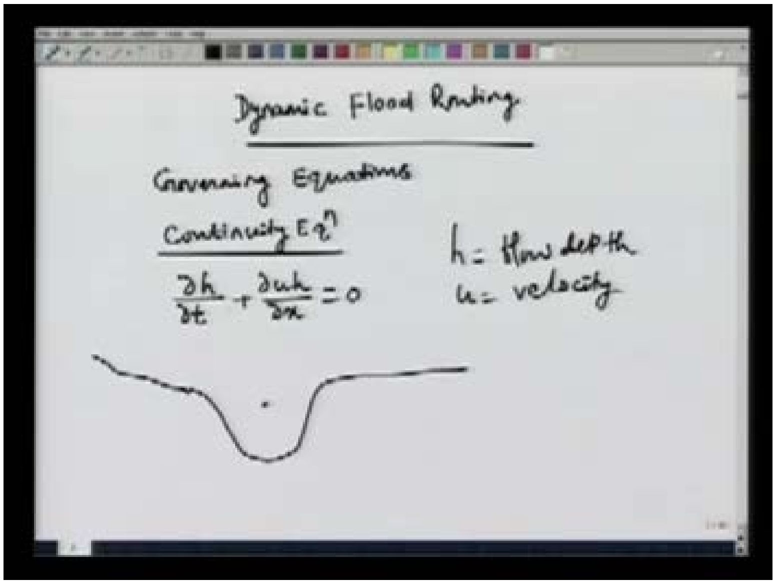
In today's class we will discuss hydraulic flood routing. In earlier classes, you have already been told about the flood routing by way of hydrologic methods. But in today's class we shall discuss the hydraulic way of flood routing.

(Refer Slide Time: 00:45)



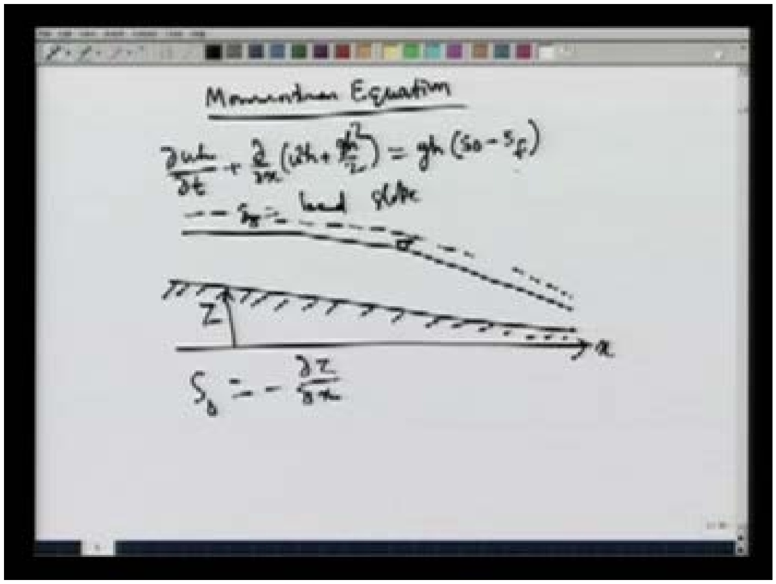
Let us say this is a channel and the flood wave is moving in this direction. This is t, this is the discharge (Refer Slide Time: 1:10). The time history of the discharge will be called the hydrograph. In this channel system I know the inflow hydrograph and here I know the rating curve. Let us say this is the downstream end of the channel. Here I know the rating curve and the stage discharge relationship. Given these conditions, I want to know the flow depth and the velocity of discharge at any other location. For example here, I want to know for a given time, the discharge or flow velocity and the flow depth. So how do we go about this? When you say hydraulic routing, it can be done in many ways. Let us begin with dynamic routing.

(Refer Slide Time: 02:21)



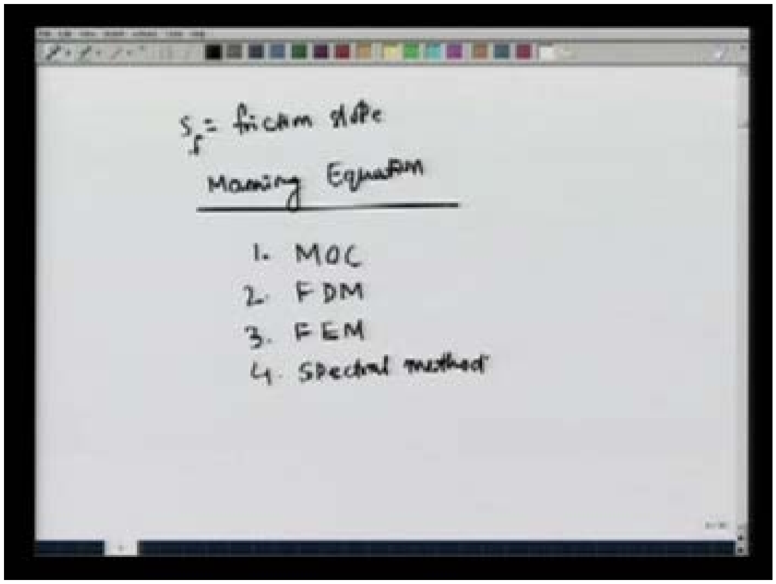
In dynamic flood routing, we use the governing equations as continuity equation for water flow and then momentum equation for water flow. If I consider the flow to be one dimensional, then I should use these equations. The continuity equation will be partial h over partial t + partial uh over partial x = 0. Remember that we have assumed the flow to be one dimensional. It means if this is the cross section of a river at any point, this will have three different components. The flow will be in three different directions. Let us say this is along the flow, this is along the width and this is along the depth. When I say the flow is one dimensional, it means that flow along the main direction at the longitudinal direction is prominent. The other two directions are not so significant. I can consider the effects of these two components of the flow velocity are minimal. I consider these directions only, so when I say this is one dimensional flow, this h is basically the flow depth and u is the flow velocity. Do not forget that we have also assumed that the bed is rigid. You know that all natural flow channels are with sediment flow but here for the sake of it, we have assumed that the bed is rigid. This means no sediment flow. This is the continuity equation for water flow.

(Refer Slide Time: 04:54)



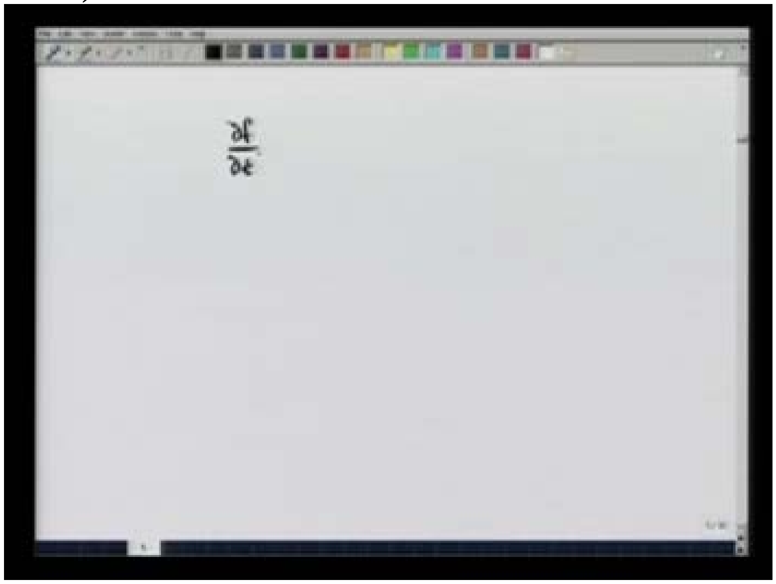
The momentum equation for water flow will be partial uh over partial t + partial over partial x u square h, gh square by 2 is equal to gh S_o S_f. Coming to different terms in these equations and as we said earlier u and h are the flow velocity and flow depth respectively, here g is the acceleration due to gravity, s_0 is the bed slope and as we are assuming one dimensional flow, s_0 or the bed slope will be the slope along the flow. It that means if this is the channel, this is slope, or in other words if I consider this to be x, this to be z, slope will be – del z del x. This is the slope or the bed slope and s_f is the friction slope. What is friction slope? Suppose this is the water surface, we will consider 3 different types of slopes. One is the bed slope which is the slope of this; one is the water surface slope, the slope of this line, another is the friction slope or the slope of the energy line. I can draw the energy line here like this. Slope of the energy line is called the s_f. If you consider the equations mathematically, these equations are hyperbolic partial differential equations and are non linear by nature. A closed form analytical solution is very difficult. We can find closed form analytical solutions only for idealized cases; therefore one should go for the numerical solution. We will study here the numerical solution to this equation but before that let me explain you this s_f, how it is estimated.

(Refer Slide Time: 07:45)



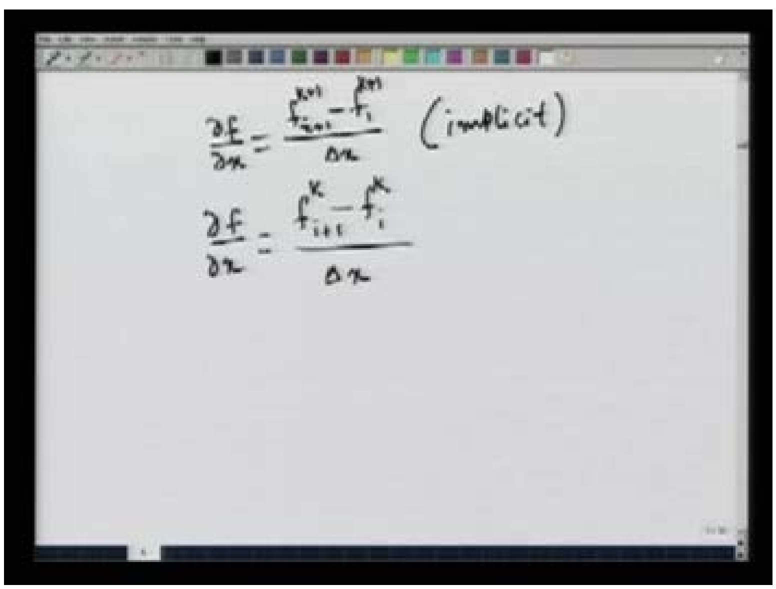
s_f is the friction slope and it should be estimated by some resistance relationship for example Manning's equation. One can use Chassis equation or Manning equation or some sort of resistant relationship. You must know that these resistance relationships are valid for uniform flow. For example Manning's equation is valid for uniform flow. It means we assume in our analysis that for unsteady flow even the uniform flow resistance relationship is valid. This is an assumption. Strictly speaking this will not be the case in unsteady flow. The resistance relationship will be different. However we assume that Manning's equation is equally valid in case of unsteady flow also. Coming back to the numerical solution, if I have to write some of the names here, then one will be a method of characteristic. There is finite difference method, and then we have finite element method, also spectral methods. There are so many variations in these methods. However, our discussion is limited to this finite difference method. The characteristic of this method is it is generally used in pipe flows, but in channel flows we generally use finite difference method. Let us consider finite difference method. I will explain here one explicit method and if time permits one implicit method. What is the difference between explicit and implicit methods?

(Refer Slide Time: 09:59)



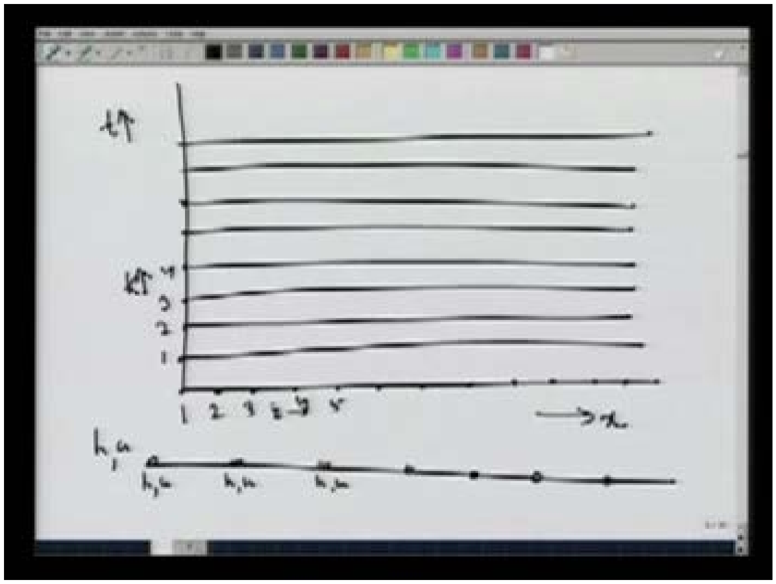
In explicit method suppose this is a partial derivative term. In the finite difference approach, what I need to do is I replace this partial derivative term with equivalent algebraic term. By doing so, I convert these partial differential equations to algebraic equations. I have a set of algebraic equations and unknowns and I try to solve the unknowns.

(Refer Slide Time: 10:27)



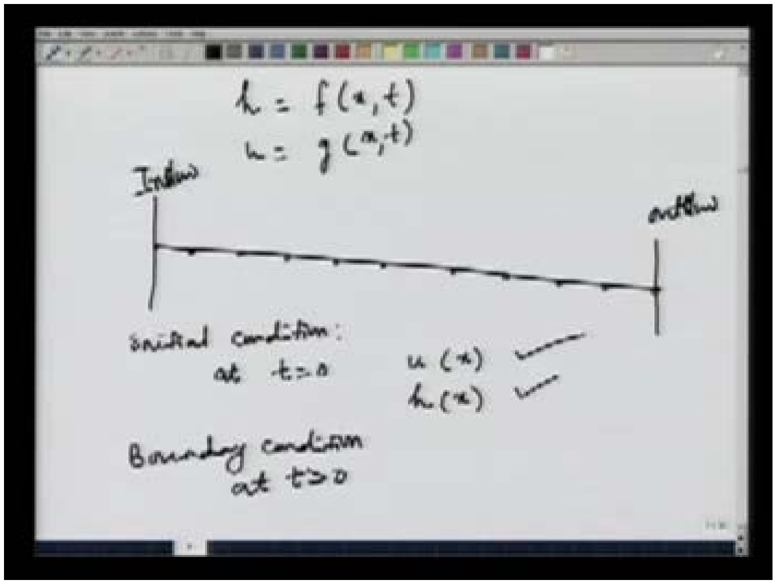
I have a partial derivative term like this, so this partial derivative will be expressed in terms of unknown time level values. For example suppose I use a forward finite difference, this is the forward finite difference, but if I look at the time levels, the time levels here are for unknown time levels, this is called implicit and in explicit, we use the same forward finite difference. But the time level will be for known time level. Here these symbols k and k + 1 represent the known time level and the unknown time level. It will be clearer when we draw the grid.

(Refer Slide Time: 11:39)



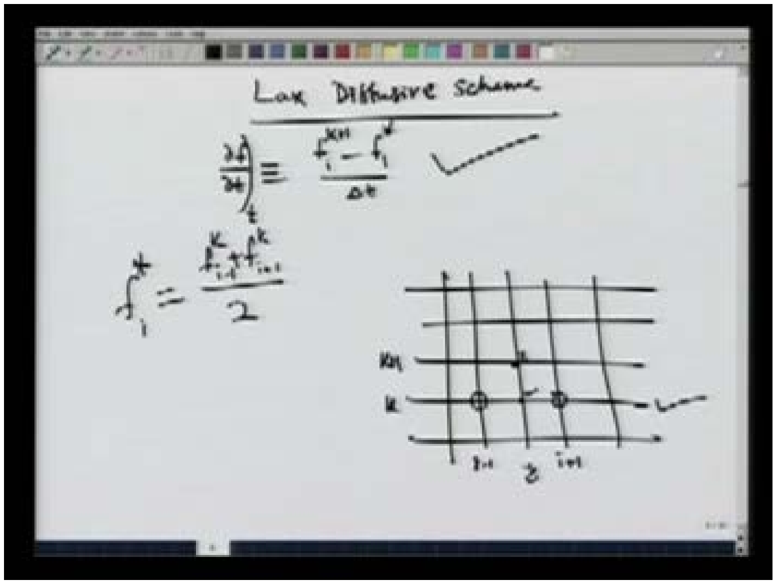
Let us say this is the finite difference grid, this is the distance and this is the time. To represent distance, I use the nodes as i's, so these are different nodes. It means different i values represent the distances and these are different time levels. Let us say I represent them through k. This grid indicates the spatial variation and the temporal variation of our problem domain. This is the spatial direction. This is temporal direction. So this x means, let us say this is 0 kilometer. This might be 2 kilometers. This might be 4 kilometers. But when I say i the designation i means some number. For example, this might be 1, 2, 3, 4, and 5, some nodes. I designate these nodes through the i values. Similarly the time nodes are designated by k values. For example k is 1 here, 2 here, 3 here, 4 here in absolute sense, this k is = 1. It may represent, let us say time is equal to 2 hours. If I am interested in knowing the variables throughout the channel, then this is the channel (Refer Slide Time: 13:23).These are the different nodes which I represent in this grid as i's, the governing equations as we wrote for the continuity equation and the momentum equation, the independent variables are x and t, the space and the time node. But the dependent variables, the unknowns are the flow variables, u and h, the velocity and the flow depth respectively. Here at all the nodes I need to find out what is h and what is u. On all the nodes here I need to know what is h and what is u. For all the nodes and also at all the time levels, I need to know how h and u vary.

(Refer Slide Time: 14:45)



h is a function of x and t varies with distance as well as with time. u also varies with distance and time. How do I define the problem? This flood routing problem will be defined by this. Suppose this is the channel, I know at a given time, the flow depth and the velocities at all the nodes. This is called the initial condition. At t = 0, for all x, I know u for all x is known and h for all x is known. So initial condition says at time t = 0, I know the variable values at all the nodes. What is the boundary condition? Boundary condition is at unknown time levels, t is greater than 0. I know the values at the boundaries or at the end nodes. As you know, the channel is divided into a number of nodes, the first node, the last node, the inflow section and the outflow section. On these nodes I should know the variables values. This is one section. This is another section. This is called the inflow or you can say the starting point of my program endowment. This is the outflow section. I know some conditions here. I know some other conditions here. From these given conditions, called the boundary conditions, my results would be affected by these conditions. Let us once again formulate the problem and we are discussing one numerical method which is explicit and the name of the method is lax diffusive scheme. We are discussing about lax diffusive schemes.

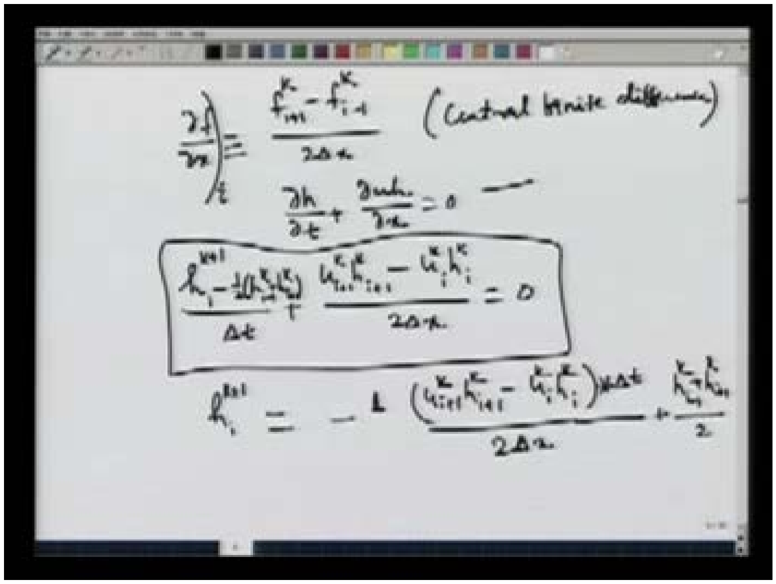
(Refer Slide Time: 16:56)



I am not repeating the governing equations. You are supposed to know these governing equations. Here the main philosophy is to write the time derivative. Let us say this is a general type of time derivative. It should be represented through (Refer Slide Time: 17:28). I will come to the star business. If we look at these expressions very carefully, this partial derivative term is made equivalent to these algebraic terms and this is for node i. That means, if you remember my finite difference grid, let us say this is a particular i and I am trying to evaluate the time derivative term at this node. For this particular node, I can write this equivalent algebraic terms and this f is the functional value. If this is h, this will be h. If this is velocity, this will be velocity at i at an unknown time level. It means if this is k, this is k + 1 if this is i, then this is unknown and this is known. Everything here is known. At this level, I do not know anything, so my objective is to find out u and h values at these nodes. At node i, del f over del t is made equivalent like this. What is f star? It is the average of the neighborhood values. Mathematically, it can be proved that if I use the value f i k, here the scheme will be unstable. What is an unstable scheme?

An unstable scheme is when the total numerical errors grow with time stepping. As time increases, the total numerical errors increase. We do not want our scheme to be stable. It means the errors should be within some limits. To fulfill that condition, we use here an averaging parameter which is because ours is a case of one dimensional problem. It will be f_i – 1 + f_i + 1 and do not forget this is for the known time level. It means if i star is the average of these two values; this is i – 1and this is i + 1. In case of a two dimensional problem, this should be made average of all the four neighborhoods. However this is a one dimensional problem and we are writing this star parameter with two neighborhood parameters. This is the substitution for the time derivative what about this space derivative.

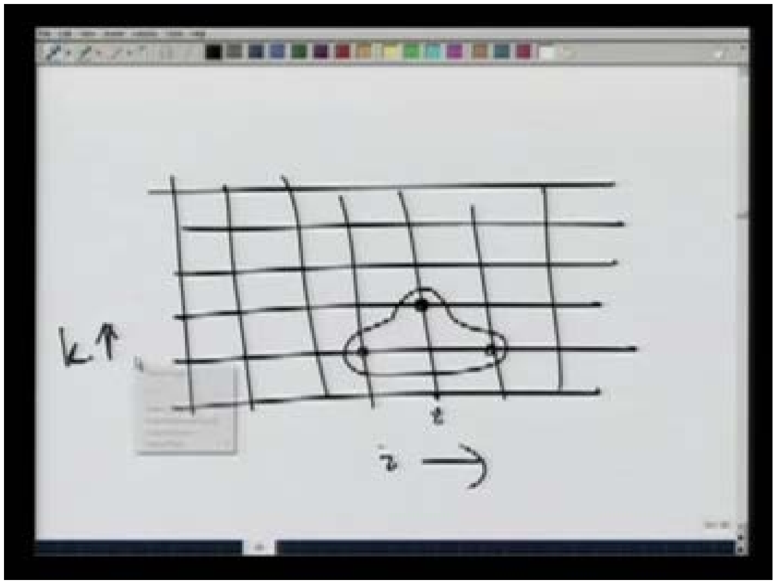
(Refer Slide Time: 20:45)



If this is a general statement for a space derivative, then I can write the equivalent algebraic expressions. This will be f_i + 1 f_i – 1 divided by 2 delta x. We use a central finite difference. In case of partial derivative, do not forget to use the known time level values here. Remember this is for node i. Once again I repeat, when we look the governing equations, we see that in the governing equations, there are partial derivatives and there are time derivatives. The time derivative terms are made equivalent to algebraic expressions by time stepping in which we use the averaging of the parameters between the neighborhood values. However in case of partial derivative we use central finite difference and since lax diffusive scheme is an explicit scheme, partial derivative will be expressed in terms of known time level values. It means here we use known time level values. I will just give an example of how to discretize or how to use these principles in the governing equations. If I rewrite my governing equation, let us say continuity equation. This is the continuity equation, in this equation I have to substitute this for the time derivative which will be h k + 1 i – h_i star divided by delta t and for this I will have h u_i + 1 h_i + 1 – u_i h_i. This should be at known time levels divided by two delta x. I substitute this partial derivative here and as we discussed the time derivative here will be equal to 0.

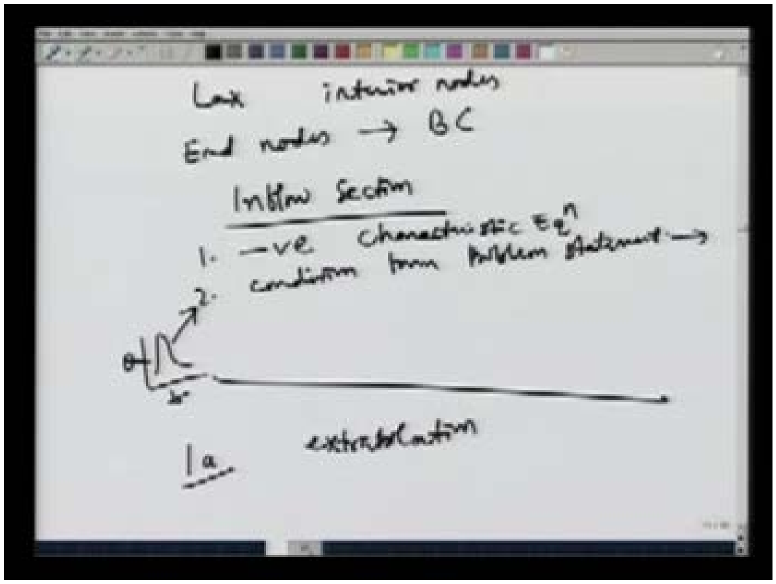
If I look at this discretized equation, (this is called the discretized equation). This is the partial differential equation. This is the equivalent algebraic equation which is the in the discretized form. This is called discretization. Here again if I replace h_i star as we discussed, it will be average of the neighborhood values. It will be half of h_i – 1 k and h_i + 1 k. If you look at this equation everywhere, whenever there is a superscript k, I know the values. Because in the numerical grid, k indicates known time level and k + 1 is the unknown time level. It means if I look very carefully at this expression, I have only one unknown time level expression. So I can write h_i k + 1 = minus here 1/2. This will be u_i + 1; h_i + 1; u_i h_i. This is with k divided by 2 delta x, then delta tu will come here multiplied by delta t and this will be + 1/2 + h_i – 1 h_i + 1. This is k. This is 2. So on the right hand side; we have all the values with superscript k that means I know these values. The beauty of this explicit scheme is that we can know the unknown time level value with the known time level values and I do not have to solve any matrix or any difficult thing explicitly. I can half the terms per each node.

(Refer Slide Time: 25:57)



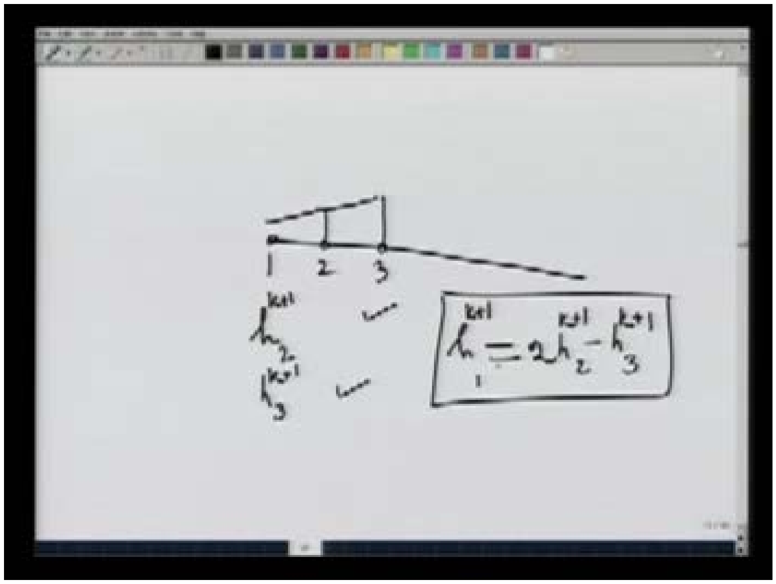
I can write this term for all the i values but when I say all the i values, if i draw the grid, this is i and this is k and if you relook the expression we wrote there, it is involving i + 1and i – 1. It means if this is my i, this will involve evaluation of h here. It will involve this and this. We have a problem now and the problem is these two values will not exist for the end nodes. For example if I am here I can have this. But there is no node here; similarly for the downstream end I do not have the node here. What we do is this lax scheme can be used.

(Refer Slide Time: 27:15)



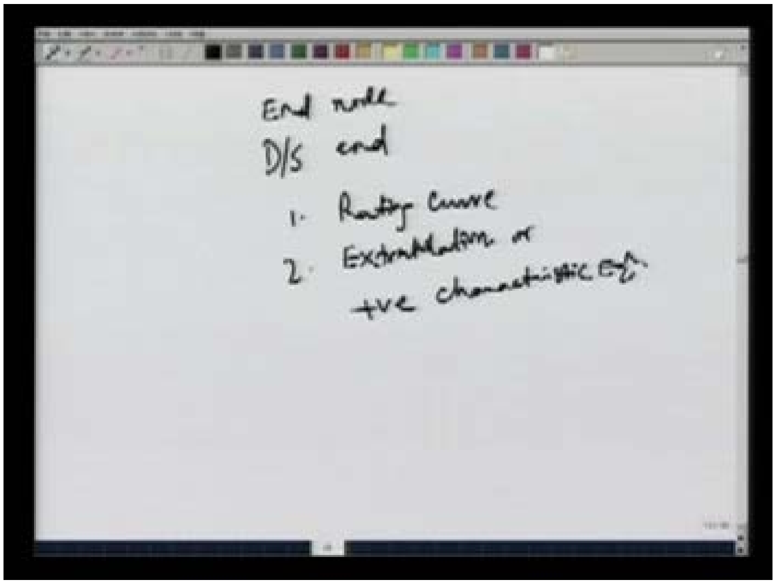
If you remember lax scheme can be used for the interior nodes and for the end nodes or the boundary nodes, we use the boundary condition. What will be the boundary condition? The boundary conditions will depend on the problem statement. When we say problem statement, in some cases it will be the inflow hydrograph for the inflow section. We have not discussed the method of characteristics but for the time being we just take it for granted that there is something called method of characteristics and in method of characteristics, we get one negative characteristic equation. For inflow section, one should use the negative characteristic equation and the condition from the problem statement. Problem statement means for example if this is the channel and you have been provided with the inflow hydrograph then this is your problem statement. So one equation is this, the other equation is this. We have two equations and two unknowns in h and u. One should be able to find out both uniquely for the inflow section. A simplified method one can use in place of this characteristic equation is you use one equation from this. So I am writing here one and in place of this negative characteristic equation, suppose you would not know what is negative characteristic equation you can go for this method. This method says you use the extrapolation. What is extrapolation?

(Refer Slide Time: 29:45)



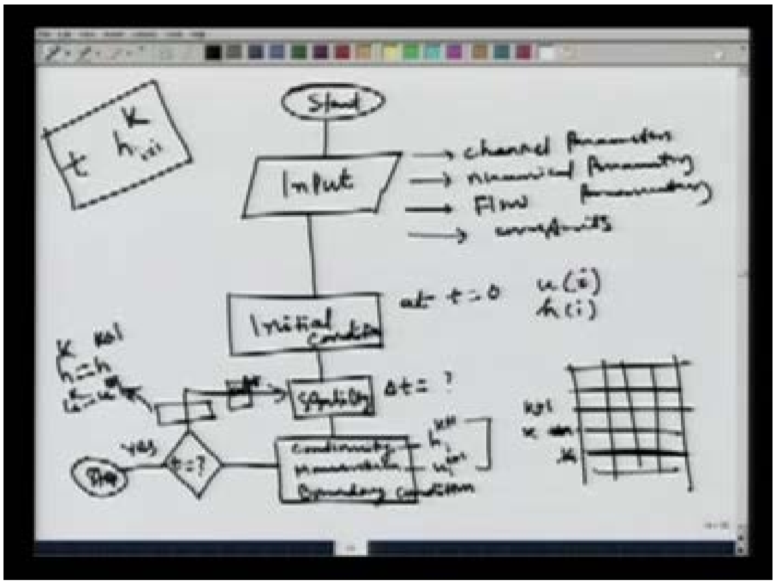
Extrapolation is if these are the nodes, I know the values. Let us say I know the h value. Here I know the h value, so you extend this line to find out h value. It means h, let us say this is 1, this is 2, this is 3, h_2 k + 1 is known. How will this be known? This will be known by the lax diffusive scheme. When we use the lax diffusive scheme for the interior nodes, I can find out h and u for these nodes. This is known. Similarly h_3 at unknown time level is known and what about h_1 k + 1? As we said we cannot use the lax scheme for this, so we will use the boundary condition. This will be expressed in terms of these two. If I use a linear curve that means I assume that the relationship between these three is a straight line. I can fit a straight line between the h values at these three nodes. It will become this plus this divided by two, which will become this or 2 times $h_2 - h_3$ k + 1. This is called the method of extrapolation. You use extrapolation method in place of method of characteristics and do not forget the condition from the problem statement. For example, if you have been provided with the inflow hydrograph, then use that condition and also this condition. These two will be sufficient to find out h and u at node number 1.

(Refer Slide Time: 31:54)



Similarly for the end node or we can say the downstream end, we have the rating curve and we have either the extrapolation or positive characteristic equation. We will discuss the flow chart of the lax diffusive scheme so that the sequential steps used in lax scheme will be clear.

(Refer Slide Time: 32:46)



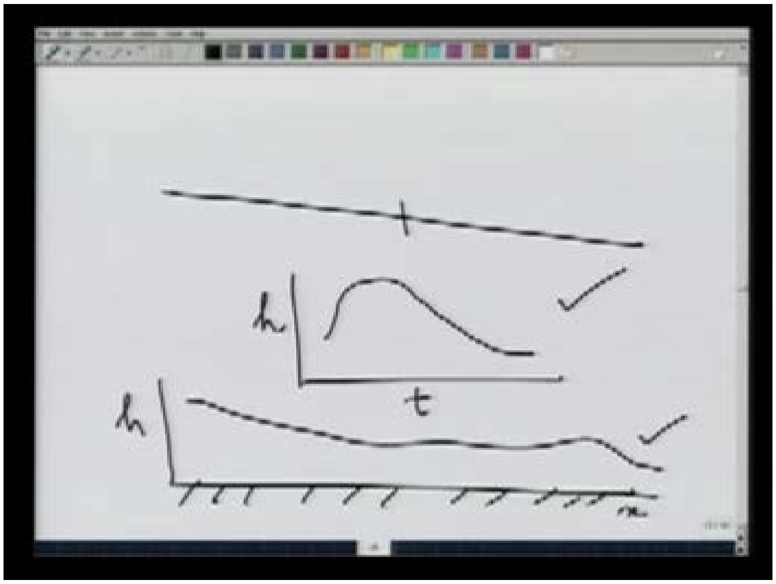
We start the program here. Then the input values are, (Input values will be different things) channel parameters such as slope, plane, it is a cross section at different places and then we have the numerical parameters. Then you may have the flow parameters and you may have the constants. For example we have g in our expression. So acceleration due to gravity g = 9.81, the

information you should put as input. Then you set the initial condition, that means at t = 0, we said u for all i's, and h for all i's. Then from this, now you have to find out what is your delta t.
I will write stability here. I have not explained what a stability criterion is. I will explain after this, because this will be a breakage in the continuity. First let us complete this flow chart and then we will discuss what stability is. For the time being, just know what the term stability means. We shall find out what is delta t. Unlike the case of implicit schemes, we have to follow certain conditions. For delta t in explicit schemes although it looks very simple, we are paying the price here. We have to follow the Courant condition to find out delta t value. We cannot use delta t values at our one. After stability, one should go for the unsteady flow condition and here you can solve the h equation of the continuity equation, solve continuity equation and solve momentum equation.

When you solve continuity equation, you will find h k + 1for all i's except 1 and the last node. Similarly here, we will find u for all i values. But you cannot get these values for node number or i number 1 and i number last. What we do is we use here the boundary condition. Please remember that these two lines are from the lax scheme. The last line or the boundary condition is not solved by lax scheme. You have to use the problem statement and common sense and general rules of the numerical method to find out the boundary condition. Then we should go for the next step but before that I should verify this. Diamond shape indicates logical statement, so I should verify whether I have reached the required time level or not. t here is verified. If I have already reached the required time level, then I should stop. If I have not reached the required time level I should repeat. Before repeating I should exchange because if this is the grid, let us say this is my k. This is my k + 1. It means this is the known time level. This is the unknown time level. Using these values I calculate these values.

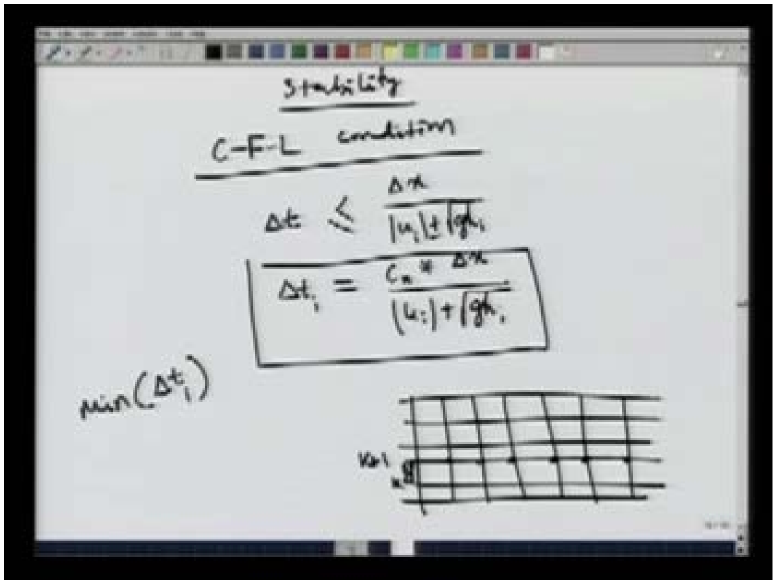
Here at the end of this, I am here. All the values are known. So I should treat this as k. It means, this becomes k and this becomes k + 1, so I should swap the variables. Here hk = hk + 1and uk is equal to uk + 1. Once again, it becomes the same thing. It means now I am at time level k which is known. For example here I move forward but before going there I should know what stability is, what delta t is. Then I solve the lax diffusive scheme, put the boundary conditions and get the answer. So I time step this that means I go to one time level and another time level like this and I will stop when I reach the required time level.For example suppose there is a flood in a river and I know that the flood takes (let us suppose) 4 hours to reach the downstream end then probably I should do the flood routing for 4 hours or may be 10 hours. The programmer has to give the time of computation, or the time level at which I want to stop my program. So that has to be given and will be given in these numerical parameters in the input file. As soon as the program reaches the time of computation, it will stop and if I want to write the results then where should I write?

(Refer Slide Time: 39:28)



It depends for example; let us say this is the channel. One sort of output will be suppose I am at a place. For a given x, I want the time variation of the hydrograph. Let us say suppose this is h, this is t. Let us say this is the flow depth. This is the time and to get the result like this what should I do? Where should I write the output? I should write the output here. Before I move here, this should be my output. Here I should write t and also hk for i. This i is fixed. Given this for any value, let us say 50, for a given distance, I want to know for each time level, what will be the variation of depth. If my objective is that for the whole domain, I should know how the surface looks like. For example suppose this is the water surface, this is the bed. So here the objective is different. This is x, this is h, we compare this and this. Here the objective is different. For this I need to do up to the end of all the computations. Before stopping I should write the output here. So depending on the objective of the user, one should use the output statement. Let us discuss the stability here.

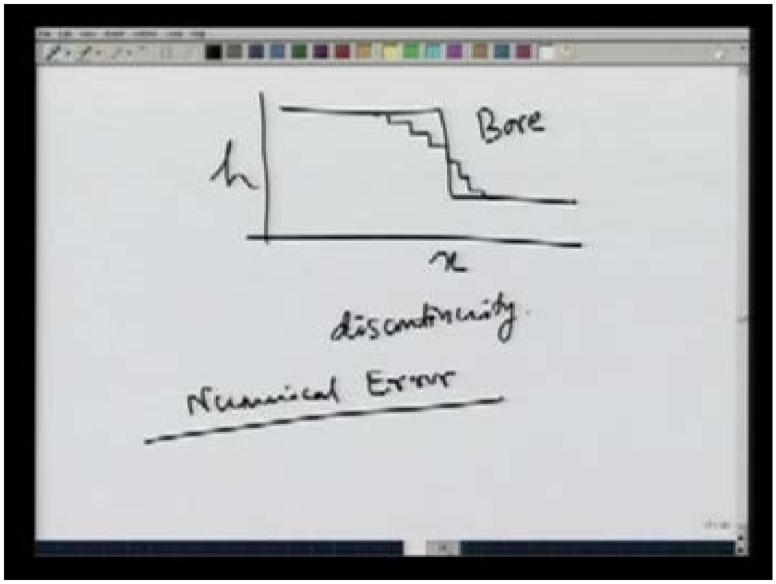
(Refer Slide Time: 41:45)



In other words if we are using an explicit scheme then we have to use a limited value of delta t. We cannot use any value of delta t. So how do we decide that? This is called Courant-Friedrich-Lewys condition, C F L condition and this condition says delta t should be less than equal to delta x divided by u_i + – square root gh_i. We multiply here, how the computer knows that delta t is less than this. To ensure that we use a factor, this is less than one. So we called that as the Courant number. We use a Courant number. It means delta t is equal to Courant number times delta x divided by u_i plus, we use plus because this will give you astringent value square root gh_i. Let me explain this. This Courant number C_n is a factor which is less than 1 and this delta x will be given by the user. u_i and h_i are known, g is acceleration due to gravity. Based on this, delta t will be calculated.

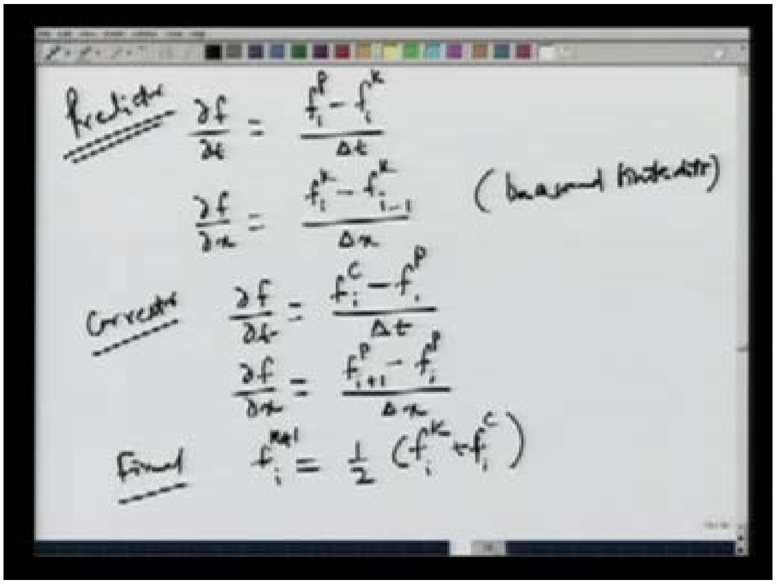
A typical value of Courant number for lax scheme is in the range of 0.6 to 0.8. One can use even higher values. One can numerically experiment what is the value of Courant number and accordingly we can decide what value of Courant number is appropriate for the given problem and once you find out delta t, please check that this is dependent on i. It means if this is the numerical grid, for each time level let us say this is the time level k + 1. To determine what will be the value of delta t, I need to find out delta t for each i and I should select the minimum of all the delta t's. We find out minimum of delta t_i. Let us say this is i so for each i, I will find one delta t value and based on the minimum criteria I will choose the minimum value of all the delta t's. So I go for the minimum of the delta t to select what will be my next time level. Once delta t is calculated, we can follow this flow chart. To find out, this is the stability step. We can move further. Please remember that this is not uniform. It means these delta t and these delta t (Refer Slide Time: 45:14) might be a different delta t and may not be equal for all the time steps.

(Refer Slide Time: 45:19)



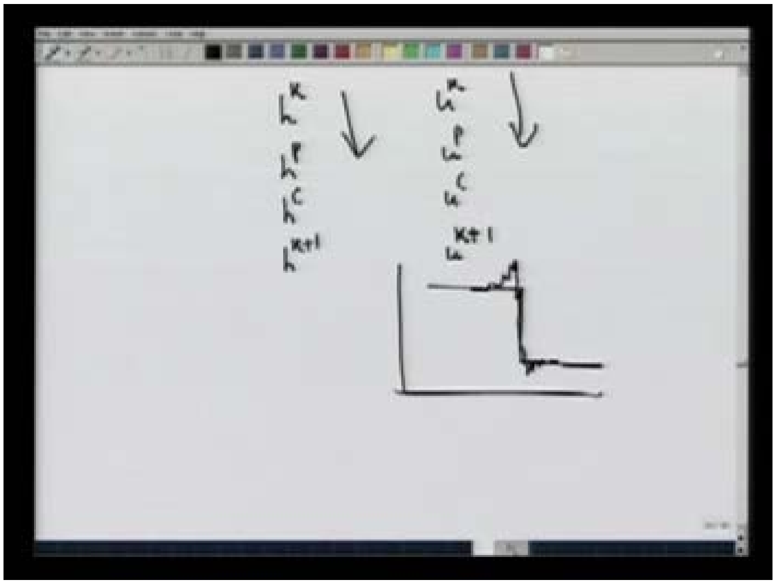
Now when you do the lax scheme, sometimes we find in flood routing, certain discontinuities like this. Let us say this is x, this is h. In flood routing, in hydraulics this is called a bore. Mathematically you can say that this is a discontinuity. So, lax scheme is not very good to capture discontinuities like this. It means strictly speaking, the numerical scheme should produce these as a result and lax scheme will give what it will give. It will indicate you the nature but there will be step like occurrences. So when we get step like occurrences, we know that this is not a very good sign. So what we do we do is we do something called a numerical error. Since lax scheme is only second order accurate in space and first order accurate in time, we find this type of errors.

(Refer Slide Time: 47:08)



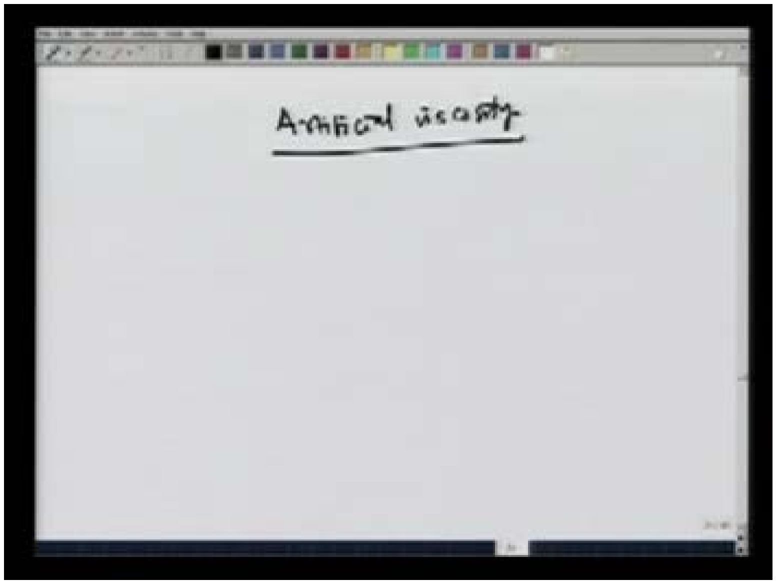
There is another scheme which is called MacCormack scheme. If you use this scheme, then step like things will not come and it can capture both very correctly. As I mentioned in the MacCormack scheme, the philosophy will be slightly different, but it is similar to lax scheme. This is a two step scheme, one is predictor scheme, and the other is the corrector step. In the predictor step, this is calculated by f, let us say p f k i divided by delta t and we also have the partial derivative terms which will be in terms of known time level divided by delta x. This is backward finite difference and this is time stepping. This is the predictor step and in the corrector step, we will have let us say c is for corrector step. This is known as the predictor step. This is used and for this partial derivative we will use the predicted values but here we have used the backward finite difference scheme. So here in corrector step, we will use the forward finite scheme. This is in terms of predicted values and this not over here. There will be a final step which we will say. What is k + 1 that means it is half of f, remember this very carefully. This is the corrector step. This is not predictor. This is at time level k.

(Refer Slide Time: 50:02)



So in the scheme of the parameters, the variables are defined in terms of let us say h and u. I have hk and uk. These two are known first. I will find out the predicted value of h. Similarly the predicted values of u from this I will find out the corrector value of h and then I will find what this is. This is the sequence. In MacCormack scheme also, it is not free from numerical errors. Any numerical scheme used will be with some error. Here we find oscillations. For example if this is the shock, we will find some oscillations here, some oscillations here, so this is also not very good.

(Refer Slide Time: 50:54)



So what is done is there is something called artificial viscosity. This is used to kill the numerical oscillations. By using artificial viscosity and MacCormack scheme, one can capture the shocks very well. To conclude today's lecture, I will say that this hydraulic way of doing the flood routing is very good. In this, we use the continuity equation and the momentum equation which are also called the (Refer Slide Time: 51:32) equations and we solve these equations by some numerical method. We have discussed 2 explicit methods. One is the lax diffusive scheme, the other is the MacCormack scheme and as I said MacCormack scheme is better because it is accurate and this scheme captures the shock very well. We will stop here.