Computational Fluid Dynamics

Prof. Sreenivas Jayanti
Department of Chemical Engineering
Indian Institute of Technology, Madras

Module No. # 02

Lecture No. # 07

Topics

Equations for some simple cases

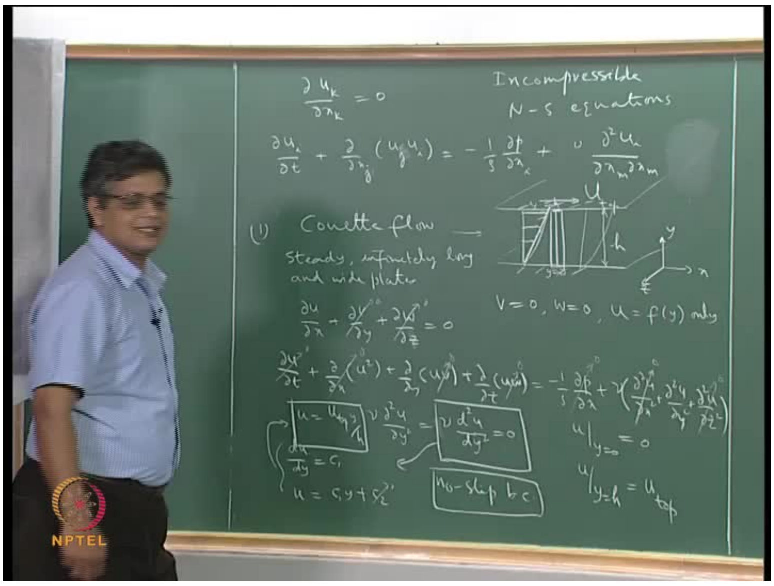
Generic scalar transport equation form of the governing equations

Outline of the approach to the solution of the N-S equations

Before we attempt the CFD solutions, let us become more familiar with the equations, and let us see if we can recognize in these things, the standard equations that we normally apply for typical cases like fully developed flow in a duct in the couette flow like that, and let us also try to understand what we mean by boundary conditions and how we are specifying it.

So, we will consider three cases - one is one-dimensional case, the other is a two-dimensional case and then third-one is a three-dimensional case, and see what the equations mean in each case and what kind of boundary conditions we specify.

(Refer Slide Time: 00:56)



So, let us start with the full form of the equations. We will write in index notation - dou u k by dou x k equal to 0 and we are considering only the incompressible form, incompressible Navier strokes equations.

So, this is the continuity equation and the ith momentum equation can be written like this. Here, we have dou u i by dou t plus dou u j dou dou by dou x j of u j u i. In this term, j is the repeated index; so, this means that j takes the values of 1 2 and 3 in order to give 3 terms coming from this equation, and if you have a repeated index which it is also a dummy index, we can as well replace this with k without changing the meaning, and it is for this reason, I have put here k and k as the dummy index there, and j here as the dummy index here, and m as the dummy index in this particular term.

You have the pressure gradient them here which is dou p by dou x i plus u the kinematic viscosity and dou square u i the same i that appears in the ith momentum equation, and the second derivative in the three directions and m is the repeated index in this particular term. So, this implies summation of three terms m equal to 1. In which case, this becomes dou square u by dou square plus dou square u i by dou x square plus dou square u i by dou y square dou square u i by dou z square and this is the gravitational term.

When we are considering flows of the single phase flow without any density changes, we can neglect this gravity and it is very easy to include this in the pressure term. So, we can, from now on, we will not worry too much about the gravitational term.

This is the equation. So, let us consider as a first example the case of couette flow. So, what we mean by couette flow is that we have flow between two infinitely long and wide plates; so, flow is taking place along this. The flow is steady infinitely long plates, long and wide plates, which are separated by a distance h, and one of the plates is moving at a constant velocity; let us say that the top plate is moving at a constant velocity capital U top.

So, this is the problem, that is, a physical problem, and let us see, if we can get the corresponding velocity profile from this equations so that we can see the equations, and then the corresponding flow situation, and what is necessary, what we are specifying, and how we can get the flow profile from this.

So, for this particular problem, let us just write down the equations. We are assuming a coordinate direction like this - a right handed coordinate frame, Cartesian coordinate frame x y direction and z direction, and we can write the continuity equation as dou u by dou x plus dou v by dou y plus dou w by dou z equal to 0.

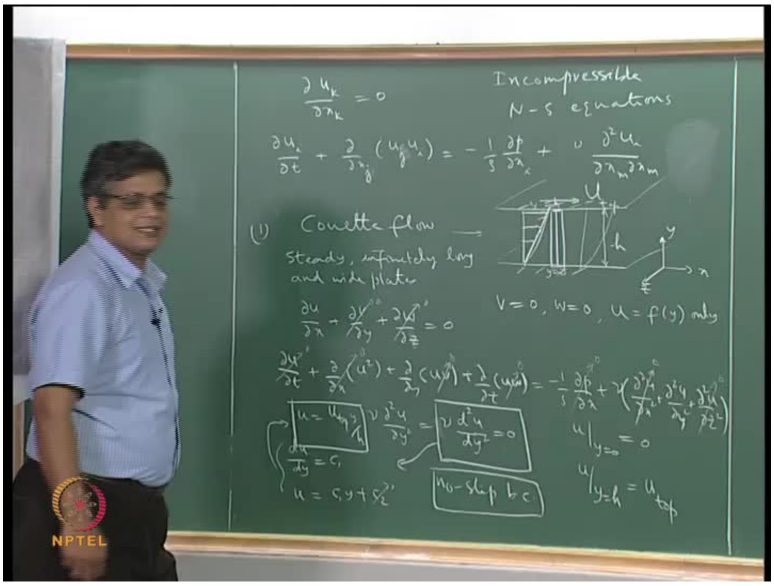
In this particular case of couette flow, we are assuming that the flow is fully developed and the flow is steady, and there is only a single velocity component which is the velocity component along the x direction. So, and under fully developed conditions, there is no pressure gradient in any direction and the flow is happening because the top plate is moving at a constant velocity. So, under those conditions, we can say that V is equal to 0 and W equal to 0 and u is a function only of y.

So, now, if you come to this part, this term is 0 because w is 0, v is 0 here, and dou u by dou x is 0 anyway, because the flow is fully developed. So, under fully developed conditions, this continuity equation does not give us any information.

Let us write down the x momentum equation. So, in which case, i is equal 1 and u 1 is equal to u and x 1 is equal to x and so on. So, we can write the equation as dou u dou t plus dou by dou x of u square plus dou by dou y of uv plus dou by dou z of uw equal to minus 1 by rho dou p by dou x plus nu dou square u by dou x square plus dou square u by dou y square plus dou square u by dou z square.

Since the flow is steady, this goes to 0, and since the flow is fully developed in the x direction, that is, there is no, if the velocity profile here is something like this, here also it is the same thing, and further down, it is the same thing. So, there is no variation in the velocity profile in the x direction; so, that means that u variation of u or u square or any other quantity with respect to x is 0 and here v is 0 and w is 0.

(Refer Slide Time: 00:56)



So, on the left hand side, everything is 0, and we are saying that the pressure gradient is also 0. The flow is taking place only because of this, and variation - we can write this as dou by dou x of dou u by dou x. So, and dou by dou x of everything is 0 here, and there is no variation in the z direction because it is an infinitely wide plate, and the plate is not moving in the z direction; so, this also 0.

So, we are left with the equation as nu d square u by d y square equal to 0, and since u is a function of y only, we can replace this with total derivative. So, this is the governing equation and we can check that this is an elliptic equation; so, that means that boundary conditions on all the sides have to be specified, all the boundaries have to be specified, and the boundaries that we are looking the flow domain. In this particular case is the one-dimensional flow domain between y equal to 0. This is y equal to 0 to y equal to h.

So, this is our flow domain, and we have two boundary conditions - at y equal to 0 and at y equal to h. So, we need to specify what is the value of u at y equal to 0 and u at y equal to h. We can also specify that Neumann type of boundary conditions or Robin type of conditions as appropriate to the flow. Now, in this particular case, we know that the plate is moving at a constant speed of U top.

So that at y equal to h, the velocity has to become U top from what is known as the no slip boundary condition. For a fluid with viscosity, we assume that the fluid, that fluid layer or the set of fluid molecules which are in contact with the solid surface will have the same velocity components and temperature and things as the solid itself. So, there is no split between, there is no velocity difference between the solid and the fluid which is in contact with it.

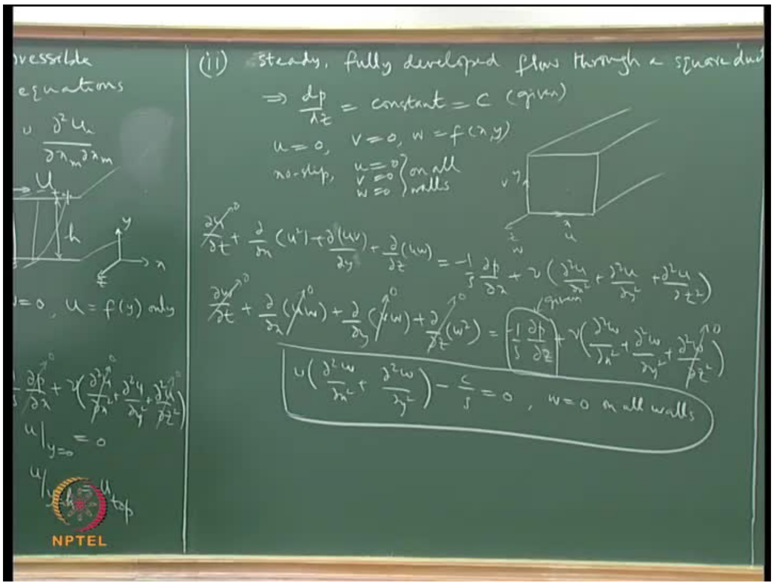
So, that means that at y equals to x, you have the solid the top plate, and at that point, the fluid also has the same velocity as that particular solid plate. In this particular case, that has a velocity of U top. By the same token of no slip boundary condition, the velocity at y equal to 0 is the other bounding surface of this particular flow domain. So, that is the bottom wall, and the bottom wall is stationary, it is not moving; so, this velocity is equal to 0. So, we have to solve this equation with these two boundary conditions, and that is the problem, and we can easily solve this.

We can say from here that du by dy equal to c 1 because d square i by dy square is 0 equal to 0 and u is equal to c 1 y plus c 2, and if we specify that y equal to 0 u equal to 0, then y equal to 0 u equal to 0 means that c 2 goes to 0, and c 1 is - when y equal to h, we have u equal to U top; so, c 1 is U top by h. So, we can write ultimately velocity profile that u is equal to U top times y y h.

This is the velocity profile that we get. So, it is a linear velocity profile. Velocity will go from 0 to U top as you go from y is equal to 0 to h. So, this particular problem is completely understood from the governing equations and the boundary conditions, no slip conditions, and we notice that the solution here depends on both the boundary conditions, that is, at y equal to 0 and y equal to h which we have used to evaluate the constants c 1 and c 2. So, in that sense if you change this one to something else, then the value of c 2 will also change, and when the value of c 2 changes, u also will change.

So, in that sense, this solution is unique for a set of boundary conditions, and it continuously changes, when you change either of these two boundary conditions. So, it is a well posed problem. So, this is the simple one-dimensional flow problem.

(Refer Slide Time: 13:34)



Let us consider a second 1 - steady fully developed flow through a rectangular duct through, let us say a duct of square cross section. So, what we are looking at is a duct of square cross section, and again, we consider the duct to be long - infinitely long - and we are considering only when it is fully developed, in the sense that if you now put x y and z like this, there is no variation of the velocity with respect to z, and under those conditions, you have a constant pressure gradient. So, this implies dp by dz is constant, and let us call this as c which is given.

So, we are given the constant pressure gradient, that is, pressure between one plane and the other plane displaced in the z direction by a certain distance, and that pressure gradient is given to be c here, and we want to know what is the velocity distribution corresponding to this, and again, we can write down these equations, and this problem is again a problem with u equal to 0 and v equal to 0 and w is a function of x and y - where u velocity is the velocity constant in the x direction; v is in the y direction, and w in the z direction. So, we can, for example, write down the x momentum equation which we have written down here.

So, the flow is steady here. There is nothing happening in the x direction or y direction in terms of the pressure gradient, and the velocity of u on all these walls is equal to 0. So, from no slip boundary condition, we will have u equal to 0, v equal to 0, w equal to 0 on all walls.

So, under these conditions, when the flow is fully developed and steady and laminar, u and v will be identically 0 throughout this particular case, and every term in it becomes 0 here. Now, let us take the more interesting case of the w momentum equation because only the w momentum velocity is nonzero. So, we can write this as dou by dou y of vw plus dou by dou z of w square equal to minus 1 by rho dou p by dou w dou p by dou z plus mu times dou square w by dou x square plus dou square w by dou w square plus dou square w by dou z square.

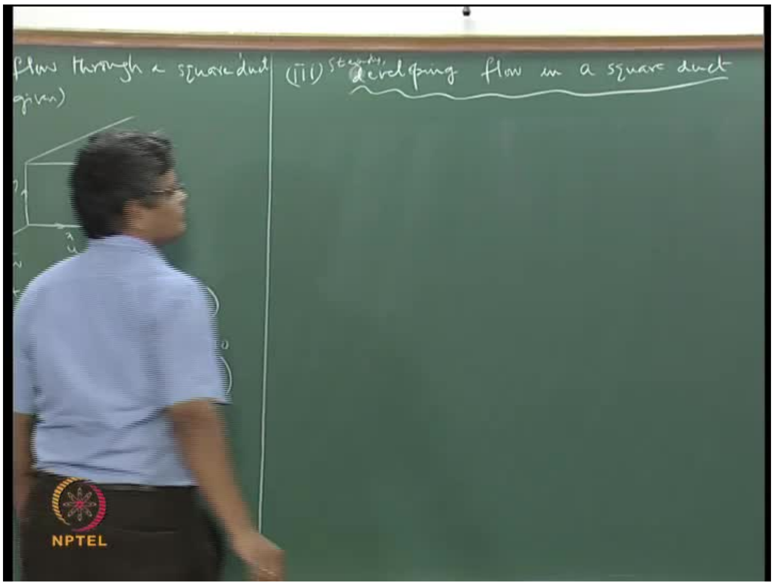
So, in this particular case, because the flow is steady, this goes to 0, and because u is 0, v is 0, these term goes to 0, and because the flow is fully developed, there is no variation with respect to z of w. So, there is no variation of z with respect to z or w square also. So, this is 0; this is a given constant, we are saying that d p by d z constant. So, this value is given. In this thing, there is no variation with respect to z; so, this thing goes to 0, but we cannot say the same thing about this, because w is it can, in fact it is 0 here; it is 0 here, but it is nonzero in between, because there is flow going through; so, that means that w has to be a function of x and w also has to be a function of y. So, we cannot apriorize say this.

So, this equation reduces to nu times dou square w by dou x square plus dou square w by dou y square minus c by rho equal to 0 - where c is given as the pressure gradient here. So, this is the equation, and this is an elliptic equation, we can see, and for this particular equation, we need to specify the boundary condition on all the bounding surface here. So, we need to say that w is equal to 0 on all walls.

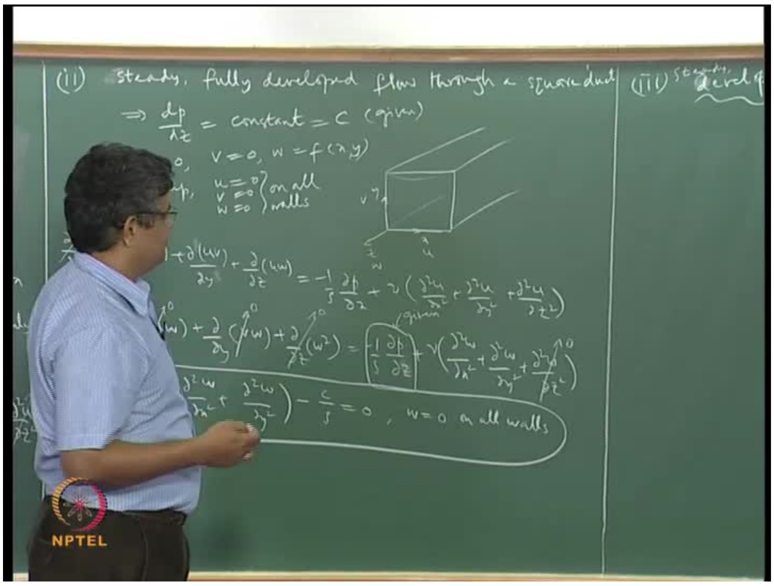
So, this is an equation in the boundary conditions, and we can see that this is nothing but the equation that we derived in the very, we solved for in the very first class. So, we actually put this as rho times mu which becomes mu here and that is equal to c, like that we have considered.

So, this is the governing equation for fully developed flow through steady duct, and the corresponding boundary conditions which are appropriate for that elliptic problem that we are considering here, and we know how to solve this. We have solved given an analytical solution here, and here we have derived CFD solution as part of the illustrative example in the first class.

(Refer Slide Time: 20:57)

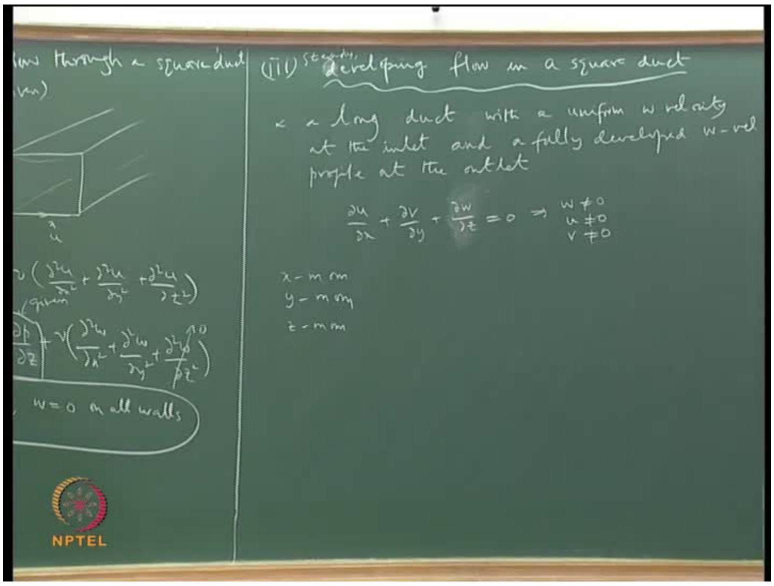


(Refer Slide Time: 21:24)



So, let us consider the third problem - again, a flow problem developing flow in a square duct. Here, we have a steady fully developed flow; we will make this also a steady. When we say developing flow, the flow is starting with something, and then, it is the velocity profile is gradually changing, and we know that in such a case, we have a boundary layer which forms from here and it forms from here, and then, eventually they merge and the flow becomes fully developed.

(Refer Slide Time: 21:55)

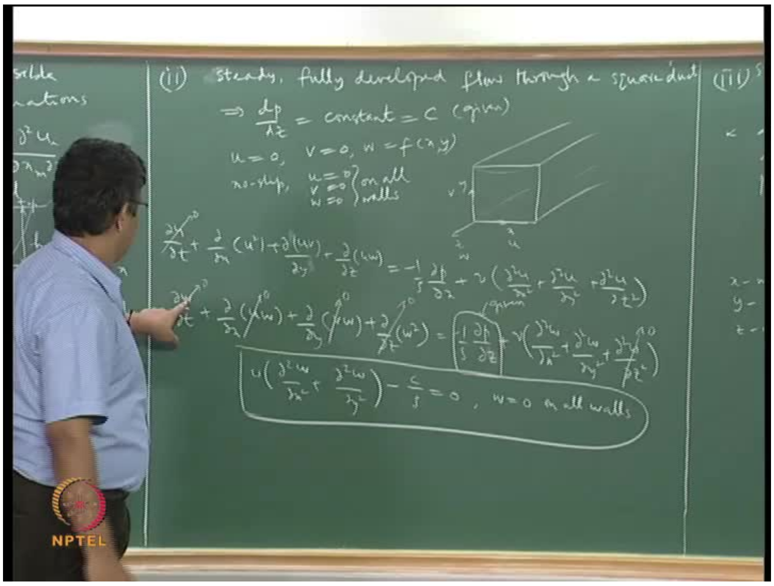


So, we want to find out how the velocity profile changes from the initial position at the entrance to somewhere at the outlet. So, we are considering here a long duct with, for example, a uniform w velocity at the inlet and a fully developed w velocity at the inlet and a fully developed w velocity profile at the outlet. So, in such a case, when the flow is not fully developed throughout, then we cannot make the assumption that u is equal to 0 and v equal to 0. So, for example, when you write down the continuity equation for this, this is the continuity equation which has to be satisfied in all cases.

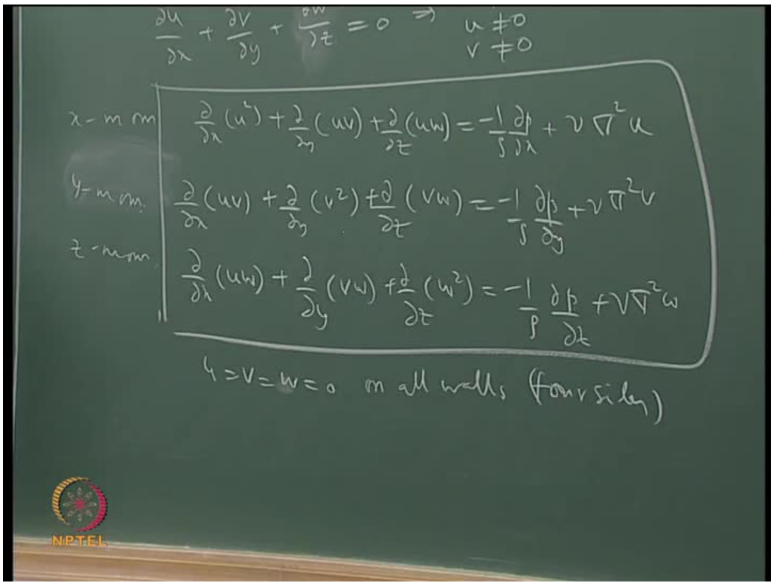
Now, the flow is developing; so, that means that the w velocity is changing with z. It is something here and then it is changing somewhere else; so, that means that this is not equal to 0; so, this is not equal to 0 so that that means that these may be also not 0 so that they can cancel out this. So, this implies that w is not 0 and u is also not 0 and v is also not 0.

So, in that sense, because the flow is developing and because the velocity profile in the z z direction is not 0 per force we have to have u and v as not 0. So, this becomes a full three -dimensional flow and we have to write down the x momentum equation which we have already written and we have the y momentum equation and the z momentum equation.

(Refer Slide Time: 24:46)



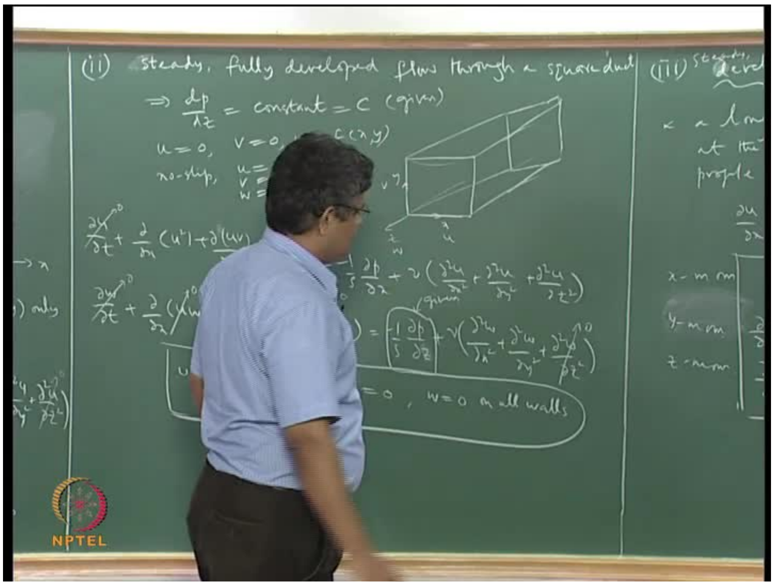
(Refer Slide Time: 24:56)



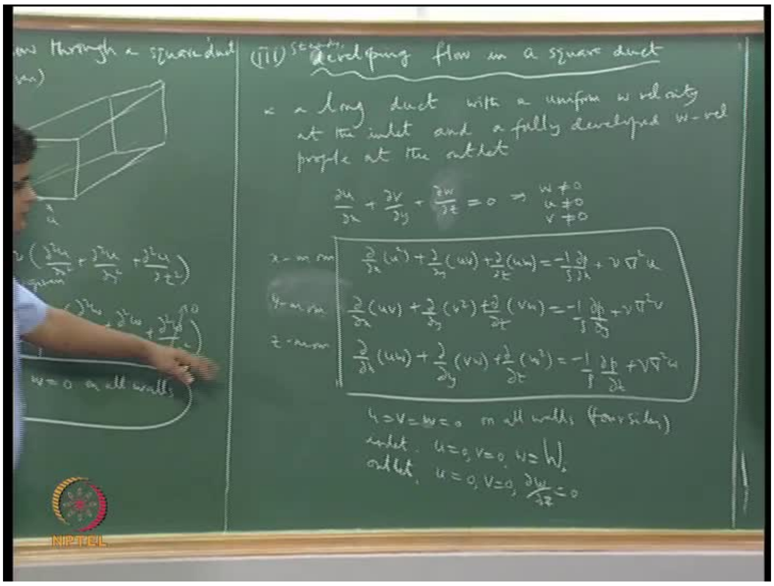
So, we will have all these terms. The only things that we can cancel out are the time dependent terms here; otherwise, we will have these terms on the left hand side, these terms on the left hand side will also appear. So, let us just write down here. So, this del square u is the Laplacian of u, and similarly, the y momentum equation we can write down, and finally the z momentum equation.

So, we have these are the equations, and in which, we cannot a prior a drop out cancel out any term because all of them are non-zero at any point within this center. They may be specifically 0 at a particular point, but they are not 0 throughout the domain; so, we cannot cancel out any of these things. So, we have three equations plus the fourth equation here and we have to solve for the fourth variables again with the boundary conditions that are u equal to v equal to w equal to 0 on all walls, there are four sides, and we also have the inlet side and the outlet side.

(Refer Slide Time: 27:06)

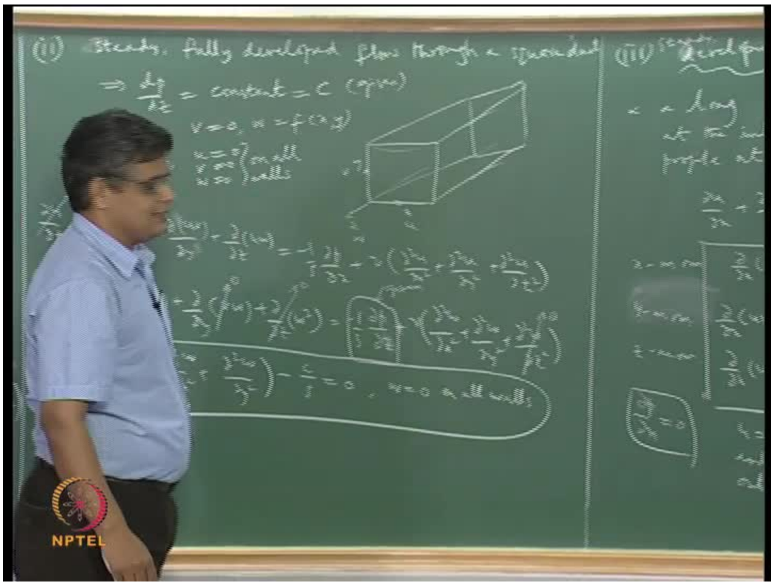


(Refer Slide Time: 27:27)



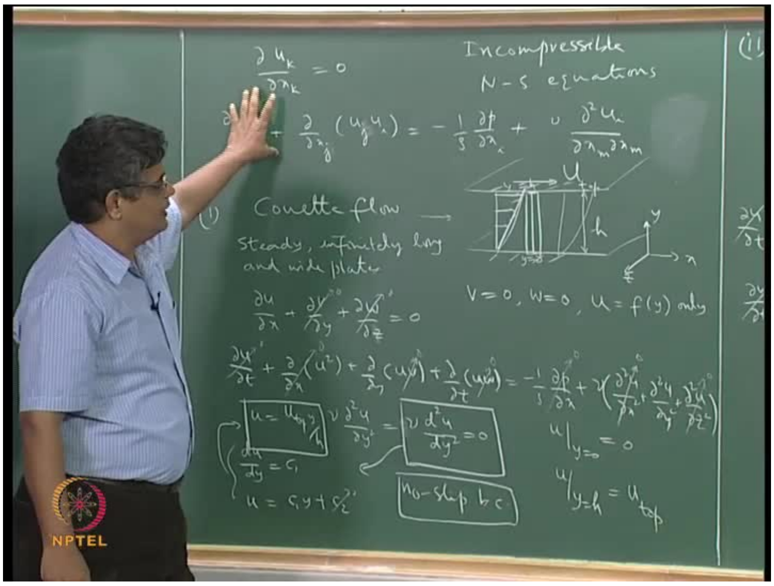
So, if we consider this to be the, so we have this inlet plane and the outlet plane. So, on the inlet, we have to specify some velocity profile. We can say that in the inlet, u is equal to 0 v equal to 0 and w equal to some w inlet which is constant; so, that means that velocity profile is velocity is constant throughout the inlet plane, and at the outlet, we can say that since the flow is fully developed, u is equal to 0 v equal to 0 and dou w by dou z equal to 0.

(Refer Slide Time: 28:43)



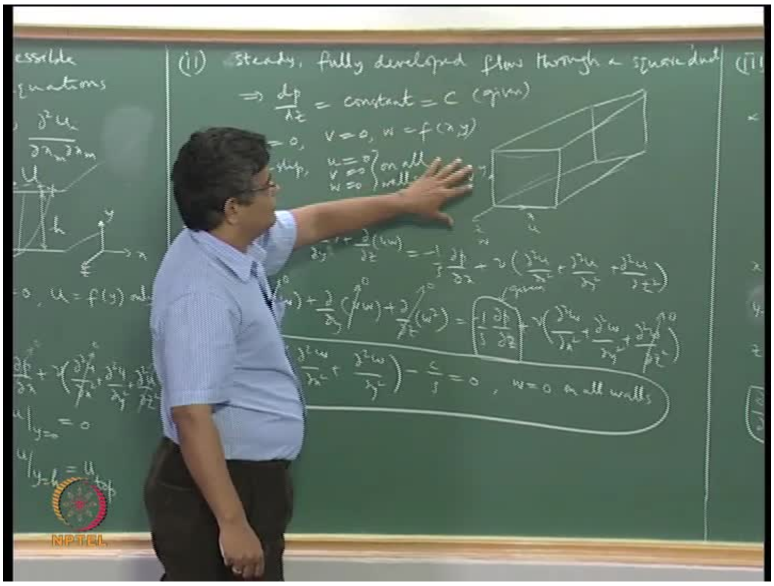
So, subject to these boundary conditions, we can solve these equations. There is still the pressure and we will see that pressure is something is a special case. We will look at the special boundary conditions later on. Typically, when we are looking at incompressible flow conditions, pressure per say the absolute value of the pressure is not important. So, pressure variation, that is, the pressure gradient that is important, and in these cases, it is implied that pressure gradient normal to the flow is equal to 0; normal to the plane is equal to 0 on all planes. So, that is the boundary condition that is usually seen to be sufficient for this, and we will also look at it later on when we look at the solution of coupled equations.

(Refer Slide Time: 29:07)

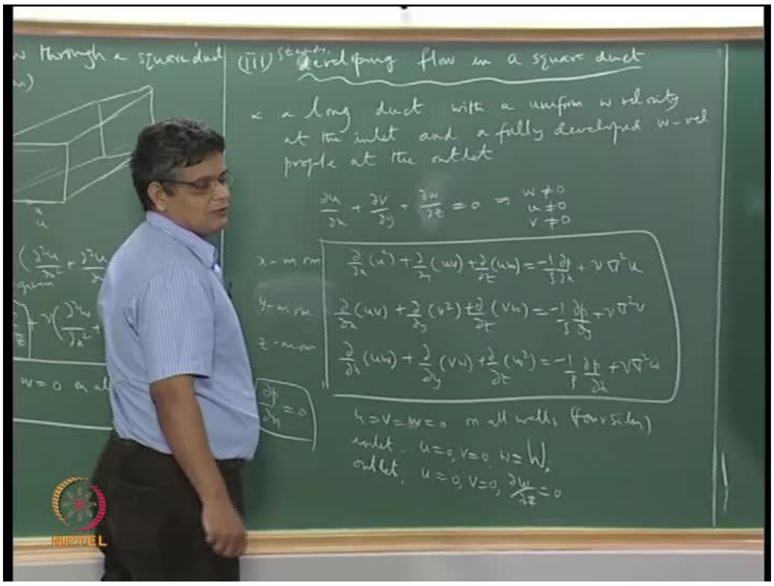


So, from these examples, we can see that the Navier strokes equations have in them all the necessary terms to look at different cases.

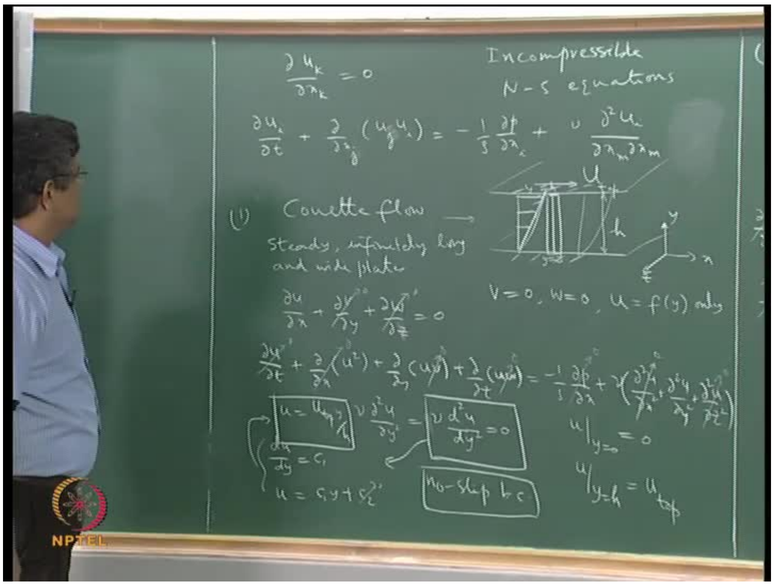
(Refer Slide Time: 29:21)



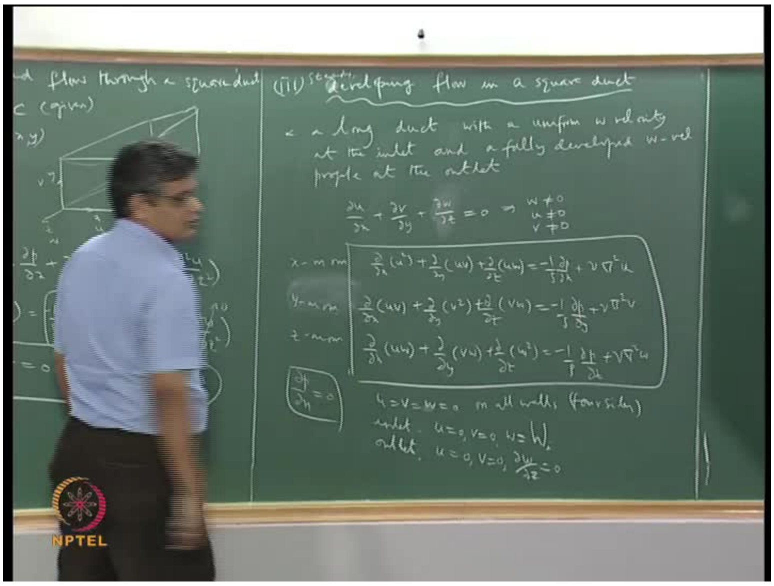
(Refer Slide Time: 29:27)



(Refer Slide Time: 29:44)



(Refer Slide Time: 30:20)



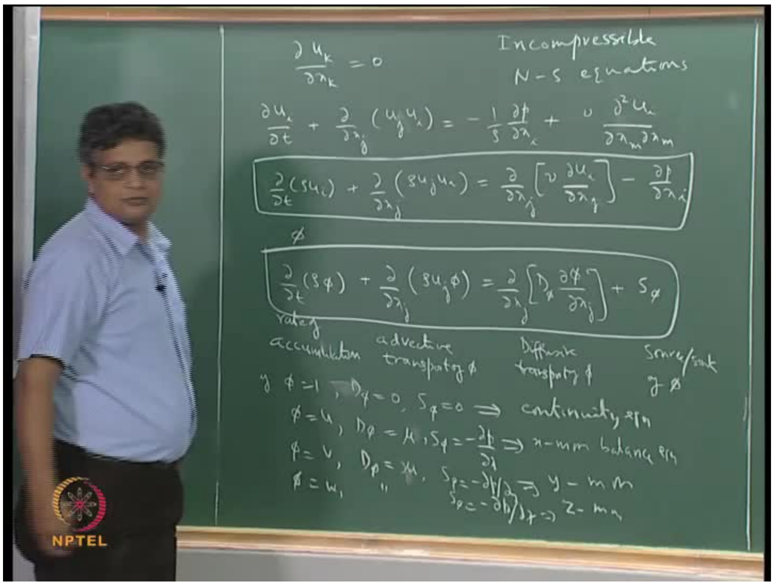
For example, the 1 dimensional case of the couette flow; the 2 dimensional case of fully developed fluid flow of a square duct, and the developing flow in a square duct, and we can also have the case of oscillating flow, for example, we can have a flow which is varying with time, in which case, we can retain the time dependent terms, and we also need to specify the initial conditions, and the boundary conditions which illustrate the condition that the flow is oscillative with respect to time. So, we, by appropriately specifying the flow domain and the boundary and initial conditions from this, we can represent the type of flow situations that we will encounter, and our objective is therefore is to solve these equation, and the equations that we solve can be of a single equation or a two-dimensional equation or a three-dimensional equation, in which, three-dimensional flow; in which, we have to solve all the four equations together.

We will see how we can solve these things, and the way that we do this is to first of all we have four equations and we have four coupled equations. For example, in the last case, we have four equations and four variables. We cannot solve either of them, any of the four independently, and that is typically the case, and for a general case, we not only have the steady conditions but we also have the unsteady conditions. So, we have variation with respect to x y z and also t. So, we have to consider how we can solve all these things together, and what will be our strategy, is to, is to delineate from these things a generic form of the equation.

We will write down these equations in a generic form, and we say that this is the kind of equation that we need to solve, and we develop a template for the solution of the generic form, and then, once we know how to solve that generic equation efficiently and properly, then we look at how to solve all the four equations together in a manner which will give us a solution.

So, the next task for us is to derive the generic form of the equation of the partial differential equation that we need to solve, and then arrive at a template at a set of CFD solution methodology which can be used to solve this generic scalar equation. Let us now put the governing equations in the form of that generic scalar transport equation.

(Refer Slide Time: 32:09)



We have the incompressible Navier strokes equation given by the continuity equation and the three momentum equations. Anticipating future changes to this, we can rewrite this equation by taking the rho here as dou by dou t of dou u i plus dou by dou x j of rho u j u i equal to dou by dou x m of mu dou u i by dou x m let us put j here - the dummy index - minus dou p by dou x I. So, we can write like this.

So, this is the momentum consideration equation and this is the mass consideration equation, and we can consider phi to be a scalar, for example, this u i is a particular velocity component, and that velocity component is a scalar. So, keeping in mind, we can ah write a generic consideration equation for a scalar phi like this dou by dou t of rho phi plus dou by dou x j of rho u j phi equal to dou by dou x j of d phi dou by dou x j plus a source term phi. So, these have certain physical interpretation. This is accumulation, rate of accumulation of phi which is also this particular thing. This is the advective transport of phi. So, we can say this is rate of accumulation phi and this particular term is a diffusion term. So, this is diffusive transport and this is the source or sink of phi.

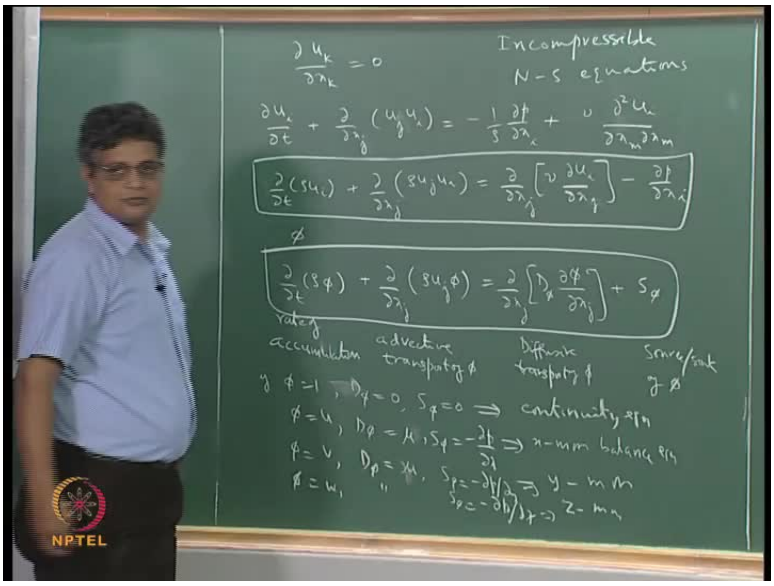
So, the rate of accumulation of the scalar phi is contributed by the advective transport, in which, phi is changing is being taken in and out of the flow domain by the flow that is happening, and by diffusion, which arises from the gradients spatial gradients of the quantity between the inside and the outside of the bounding surfaces, and any source terms that are appearing within the domain or from outside the domain like this, and if we consider this, if we consider this to be a generic scalar transport equation, unsteady scalar transport equation with advective transport given by the fact that it is a flowing system, and diffusive transport given by the fact that fluids have diffusivity of momentum heat and mass and all that plus a source which can arise in the case of momentum from external force field like the pressure, and in the case of heat by having a heat source or heat sink or in the case of mass transfer by some other things, and in the case of chemical reactions by the rate of chemical reactions, so, there are so many ways by which source or sink may be present, and this is the generic scalar transport equation and we note that if phi is equal to 1 and diffusivity of phi is equal to 0 and s phi equal to 0, then we get the continuity equation.

So, when phi is equal to 1, we have 0 by d t, and here, we will have d by d x t of rho u j. So, that is our dou by dou x of rho u plus dou by dou y of rho v plus dou by dou z of rho w d phi is equal to 0; this term goes to 0 and s phi equal to 0; this term equal to 0. Then we recover from this generic equation, the continuity equation.

If phi is equal to u and d phi is equal to mu, so the diffusivity that is appearing here, and s phi equal to minus dou p by dou x, then we recover from this the x momentum balance equation. So, you put phi equal to u u here, you get dou by dou t of rho phi plus dou by dou x j of u j u and j is the repeated index. So, you get for j equal to 1, you get dou by dou x of rho u square; when j equal to 2, you get plus dou by dou y of vu and dou by dou z of rho w u.

So, the three terms that appear here, and here this is d phi is equal to mu. This d phi should be equal to mu; this is the dynamic viscosity. So, you get dou by dou x of mu dou u by dou x. So, that is mu dou square by dou x square. Similarly, dou by dou y of mu dou u by dou y dou by dou z of mu dou u by dou z. So, the three terms associated with the Laplacian will come here, and s phi is minus dou p by dou x which is the pressure gradient, and when phi equal to v, d phi equal to equal to mu, s phi equal to minus dou p by dou y, we get the y momentum equation, and when phi is equal to w, d phi is equal to mu, and s phi is equal to minus dou p by dou z, we get the z momentum.

(Refer Slide Time: 32:09)



So, from this, we can say that this equation here for different values of phi and different values of d phi and source term can represent either the continuity equation or the three momentum equations, and it can also represent other equations like the energy balance and the species balance equations which will come later on when we were looking at turbulent reacting flows, turbulent reacting non isothermal flows.

So, in that sense, this is the generic partial differential equation of second order. We have second derivative coming in this term. This sharing certain special features of advective transport and diffusive transport which needs to be solved and this is the generic transport equation that we would like to solve in the case of CFD.

Now, we need to also we would like to consider here the special form of this particular transport equation in the form of advective transport. Advective transport is transport essentially in the direction of flow. So, this is what that brings in hyperbolicity to the

Navier strokes equation, because when you have a flow, it has a certain direction of flow and the velocity of the flow. So, that is what you mean by certain wave like solution.

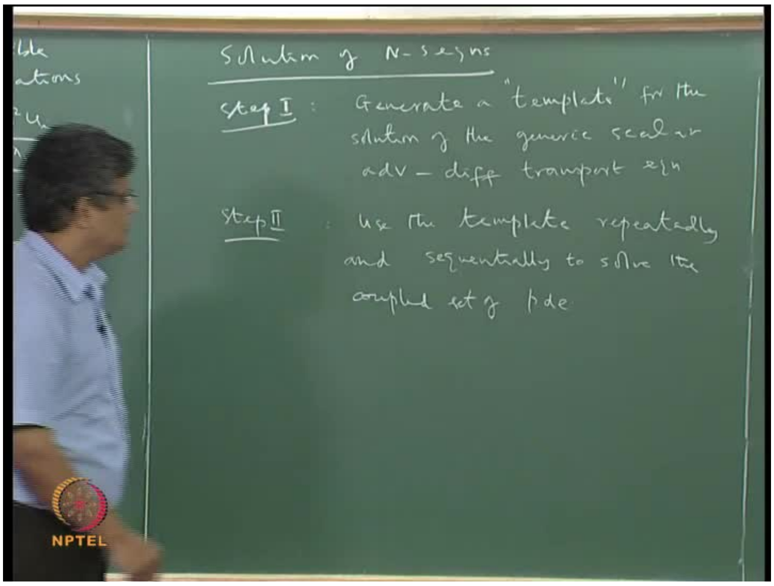
So, this is what that brings in hyperbolicity to the problem and this is the diffusive transport and diffusive transport is directionless and in a way velocity less. So, this is what brings in ellipiticity to the Navier strokes equation.

So, the generic form of the scalar transport equation has hyperbolicity coming from the advective transport equation, and then, ellipiticity that is coming from the diffusive term, and a parabolicity which is coming from the rate of accumulation term.

So, if you have a case where the diffusive transport is negligible, then you have a parabolic hyperbolic type of equation, and in the case where the advective transport is 0, then you have a parabolic elliptic type of equation.

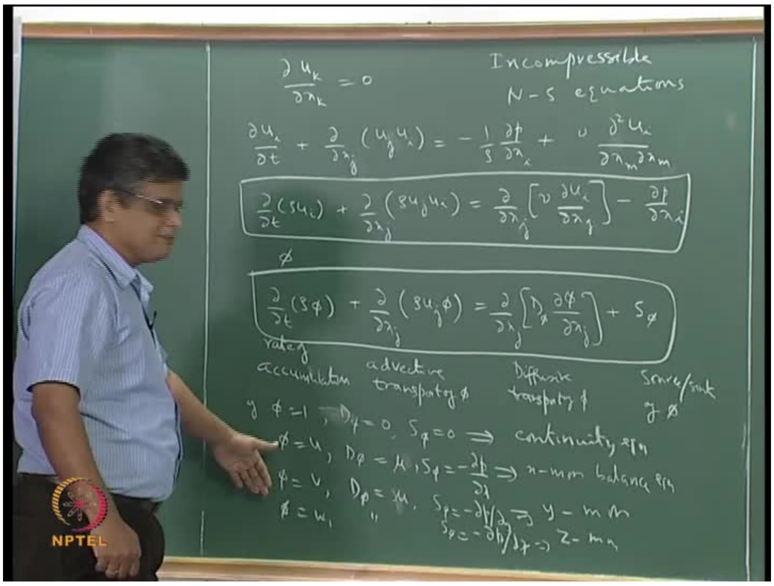
So, in the case where both are equally predominant or non negligible, then you have a mixed type of equation. So, the equation that we are trying to solve here does not strictly belong to the general classification and also noting that we have coupling with the other equation means it is very difficult to say what exactly the equation is in pure one-dimensional cases. We can attribute to either of any of the three things, but in the general case, we have only tendency of hyperbolicity or tendency of ellipiticity as a predominant case, in case advective transport is the dominating thing or a diffusive transport is dominating thing and we must keep this in mind when we look at the well posedness and all that.

(Refer Slide Time: 43:23)



So, now, our task is to solve this equation. So, we would like to first solve to able to solve this equation, and then, we will see how we can solve the coupled equations. So, we break up the problem of solving the Navier stroke equations in two steps: Step 1 is generate a template for the solution of the generic scalar advective diffusive transport equation, and step 2 is to use the template repeatedly and in a way sequentially. This is the popular method to solve the coupled set of partial differential equations.

(Refer Slide Time: 45:12)



So, first we generate the template for the solution of one equation, and the same equation for different values of phi d phi and s phi will take different things. So, the same method will be used to solve this and then this, this, this, in a sequential way, and then, this is done successfully and iteratively so that we can solve all the equations together to take account to the coupling. (Refer Slide Time: 45:18)

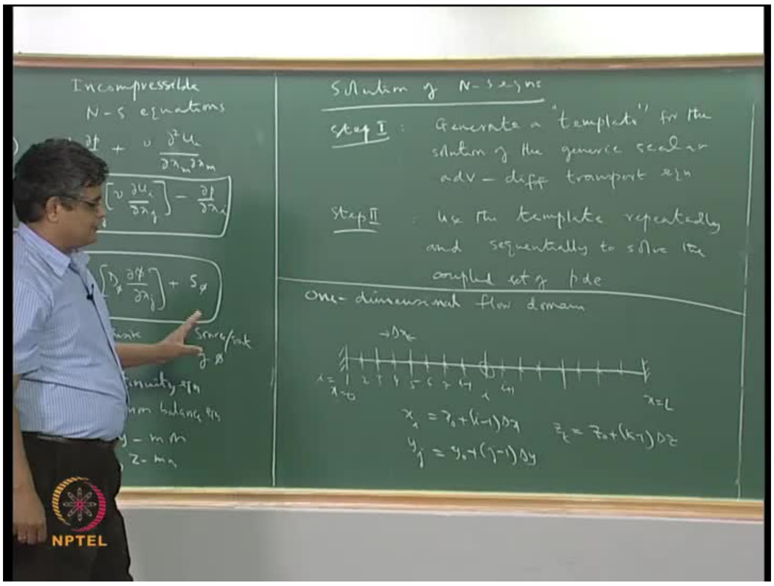
So, the first objective is to do the first step to generate the template. Now, in generating the template, we know that the steps of the CFD are first form to do the spatial discretization, and then, on that spatial discretization, we discretize the governing equation, wherein, we seek a solution to the governing equations not in the exact form, in a form which is approximate and which is approximately satisfied at each of this or grid nodes or within the sense of each of those things, and by applying this approximate form at each of these grid nodes, then we get a coupled set of algebraic equation which we want to solve.

So, the first step in the solution of the generic transport equation is to do the spatial discretization. Now, when we generate the template, we do not want to bring in all the complexity. So, we will take the simple case where we assume that the domain that we are interested is something that can be easily be discretized.

So, we are talking about a regular domain. For example, a one-dimensional domain or a two-dimensional domain which can be fit in a rectangular coordinates like what we have seen or even in a cylindrical coordinate or spherical coordinates in one of the standard things, where the discretization is very simple, because we can then consider x equal to constant line and y equal to constant line, and then identify the control volume. The discrete elements or the tiles which make up the control volume as the points of intersection of this constant x equal to line and constant y equal to line. So, we take this particular approach for the time being.

So, we consider only simple geometries which we can be discretize simply, and on this simplified discretization, we look at generation of the template and we will see that there are certain principles that we have to adhere to in generating a template which we can solve the generic transport equation, and these principles when embedded in the solution method methodology will ensure that the resulting solution of the approximate form of the equation will not only give us satisfactory solution but one which is capable of approaching the real solution the exact solution when you go to extremely fine grid space, that is, when you have large number of grid points spread throughout the domain, then this equations will be capable of approaching the exact solution.

(Refer Slide Time: 49:05)



So, we want to have that kind of template which will asymptotically, in the asymptotic case which will give us those principles. So, we would like to highlight those principles which have to be embedded into the solution algorithm in order to get that kind of desirable solution methodology, and in order not to complicate all that by taking a very complicated chip. We take the simple case of flow geometry, and for the time being, we will consider a one-dimensional flow.

One-dimensional flow, in which, x equal to 0 to x equal to l is the overall flow domain. We have a boundary condition given at x equal to 0 and a boundary condition given at x equal to l and also the initial condition as demanded, by the, by the solution and by the well posedness requirement, and we consider this to be divided into uniform segments. So, this is divided into four segments and this is divided in to eight equal segments and this into sixteen equal segments like this with a spacing of delta x which is constant and, the, these points here are indicated by an index i.

So, when you say that i equal to 1, this second point third point 4 5 6 7 like that. So, in order to indicate a general point, for example, this, we say that this is i, and the immediate point to the right of this will be i plus 1 and the immediate point to the left of this will be i minus 1 and so on i plus 2 i minus 2 and so on. And in that sense, for a given point i, we know the neighboring points on either side, and we know that x corresponding to the x location corresponds to any i from this is equal to 0, that is, the value at x equal to 0 in this point, that is, in this particular case, we have taken x equal to 0 plus i minus 1 times delta x.

So, this gives us the value of x at the ith grid point. Similarly, if you have a two-dimensional thing, in the y direction also, we can also do the similar kind of thing so that we can say y j is equal to y 0 plus j minus 1 times delta y, and in the three-dimensional rectangular coordinates with uniform grid spacing, we can also have z k equal to z 0 plus k minus 1 times delta z.

So, this indicates the x y z of a particular grid point which is located at the intersection of x equal to constant line, y equal to constant line and z equal to constant line. So, in that sense, and when you make that sort of grid, which is, for example, possible for a regular flow domain, which can be which is encompassed by lines of planes of constant x, constant x, constant y and constant z.

Then we can make up a simple grid like this and these identify the grid points, and the grid points, the grid points can be used as the vertices to construct a set of bricks in of 3 dimensional cases or tile in three-dimensional cases or line segments in one-dimensional cases. So that when we you put everything together, you get the entire flow domain from x equal to 0 to x equal to x l and y equal to 0 to y equal to y l and z equal to 0 and so on like that.

So, that kind of simplified discretization of the flow domain is what we are considering in this, and on that simplified flow domain, we will look at how to discretize this. So, when we discretize this, there are several methods - we have finite difference method and finite volume method finite element method and even spectrum methods and so on.

In this particular course, in generating the template in order to understand the additional principles which are necessary and specific to computational fluid dynamics; in order to illustrate those things, we take the simplest approach possible which is the finite difference. We make use of finite difference method for the discretization of this scalar equation on this simple grid here to arrive at the discretized equation which we would like to solve, but before we solve those discretized equations, we would like to analyze them to ensure that by solving these equations, these discretized equation, we will get satisfactory solution.

So, that is what we are going to do in the next class. So, we will start with this equation and this simplified grid spacing, and develop systematically the principles by which we can find finite difference approximations for each of this terms.