Computational Fluid Dynamics

Prof. Sreenivas Jayanti

Department of Chemical Engineering

Indian Institute of Technology, Madras

Dealing with complexity of physics of flow

Lecture No. # 31

Topics

Overview of the approach of numerical solution of NS equations for simple domains

Introduction to complexity arising from physics and geometry

We have come a long way from the early days, when we were deriving the equations. We have looked at the equations - the conservation equations - to be followed to be obeyed by any fluid motion and out of these equations representing the mass conservation and momentum conservation, we formulated the generic scalar transport equation. And then, we looked at how to discretize the generic scalar transport equation, in terms of writing finite difference approximations for the each of the derivatives. And then, we also saw that simple application - simplistic application - of the finite difference approximation, may not give us a proper solution; that is, in some cases we may get instability; some cases, we may get inconsistency, and so on.

And then, we formulated therefore the method of analysis of the discretized equation in terms of three properties, which is consistencies, stability and convergence. And having looked at these things, then we came up with a template for the solution of transient scalar transport equation, which would give satisfactory solution under usual circumstances. And then, we looked at what is how we can apply this template for the solution of the coupled equations, representing the continuity equation and the three momentum equations; and there we found that, for incompressible flows which is of main interest in this particular course, the coupling is not so straightforward; in the sense, that the pressure could not be easily derived from the continuity equation, where therefore looked at several methods, several approaches, to coupling all the four equations together, so that we would get an overall calculation procedure.

And then, we said at this calculation procedure would involve the solution of matrix equations; each matrix equation representing the conversion of an original conservation equation in the form of partial differential equation into the same conservation equation in discretized form, applied to all the grid points for which we need to have the solution variables.

So, this A phi equal to b is a type of linearized algebraic equations, that we had to solve repeatedly; and therefore, we looked at several methods at, which would, which could be used for the solution of these things. And then, we looked at a variety of methods, some direct methods like, the gaussian elimination method and L u decomposition method, and basic iterative methods like, the Jacobi method and Gauss seidel method and successive over relaxation methods, which could be used for the solution of this. And then, we also looked at some specialized methods, which would combine the advantages of both the direct methods and the iterative methods to come up with improved efficiency of the solution of A phi equal to b.

So, therefore, now, we can say that, we know how to do CFD for a general case, and what is the what is the way that we do it. We write down the governing equations, we write down the continuity equation and the Navier stokes equations. We have so far considered Cartesian and coordinate frame, and let us stick to that. So, once we write down the governing equations, then we take, we discretized the domain and identify the points at which we would like to get the solution.

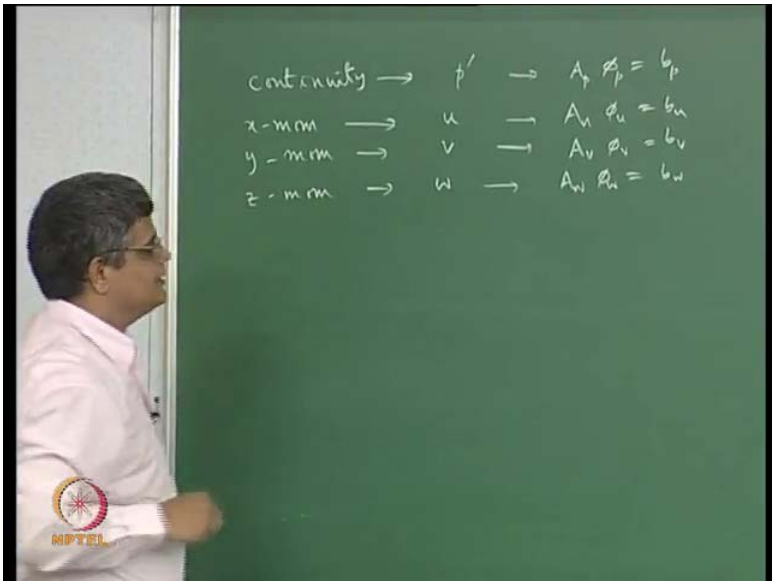
So, these are the grid points at which the variables are to be evaluated. Now, at where these grid points lie, depends on the way that we do the discretization. We have looked at the specific case of uniform discretization in the x direction and y direction or x direction and t direction. And at each of these grid points where the equation, the variable has to be evaluated, we make use of the corresponding conservation equation; and we substitute for all the derivatives that appear in the corresponding conservation equation, finite difference approximations of a desired accuracy, and come up with a discretized form of the governing equation. And then, we analyze the discretized form of the governing equation to check for stability and consistency.

So, once we are satisfied that consistency and stability conditions are being met for this particular discretization scheme, then we will have an equation of the form A phi equal

to b solved, in case we are using an implicit method or in case we are dealing with a steady state equation. So, then, we can choose from the suite of methods that we have described, to choose one particular method which we would like to apply for this, A phi equal to b and then we would solve it.

Now, in a typical case, we have four equations to solve for the u, v, w and p; therefore, we have an overall algorithm, which can be briefly put up like this.

(Refer Slide Time: 06:13)

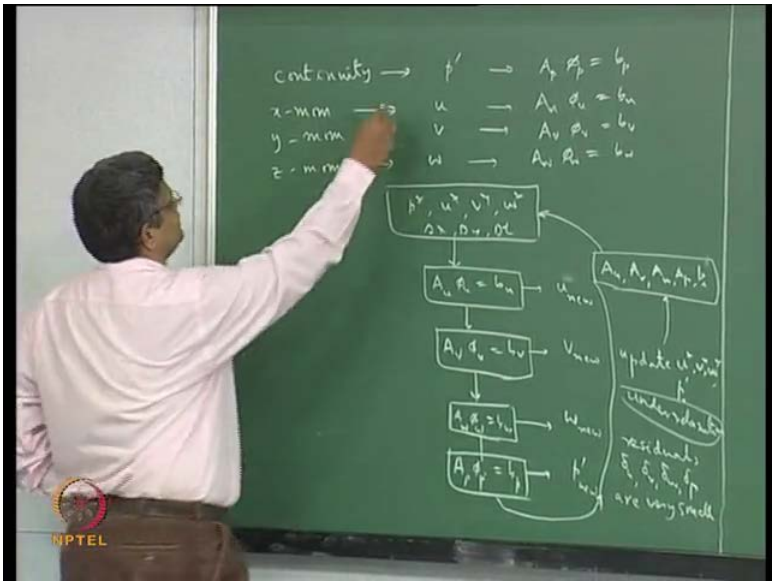


So, we have the continuity equation from which, for example, we solve the for the pressure correction in the case of the simple method, and we have the x momentum equation, which we solved, which we used to solve u, the y momentum equation for v and z momentum equation for w. Now, we have the corresponding form of conservation equation, which will give us the pressure correction and u and v and w, at the appropriate points. And we note that, we made use of the staggered grid approach and the form of the pressure correction, that we have already described. So, each of these can be put in the form of A p phi p equal to b p, and this A u phi u equal to b u, this as A v phi v equal to b v, A w phi w equal to b w.

We also make the point, that in deriving these equations, we make use of the boundary conditions and we treat the boundary conditions in the appropriate way; for example, if we have a Neumann boundary condition, we make use of the Neumann boundary

condition to derive the equation for the particular variable, at that particular point. If it is a interior point, we make use of the conservation equation, but if it is a boundary point, we make use of the corresponding boundary condition to derive the equation; and thereby, we derive the discretized equation like this. Now, each of these can be solved to give us, p, p prime, and u, v, w, but we know that, the coefficients that are coming here involve the values of the variables of the other things, and therefore, we do it in an iterative way.

(Refer Slide Time: 08:38)



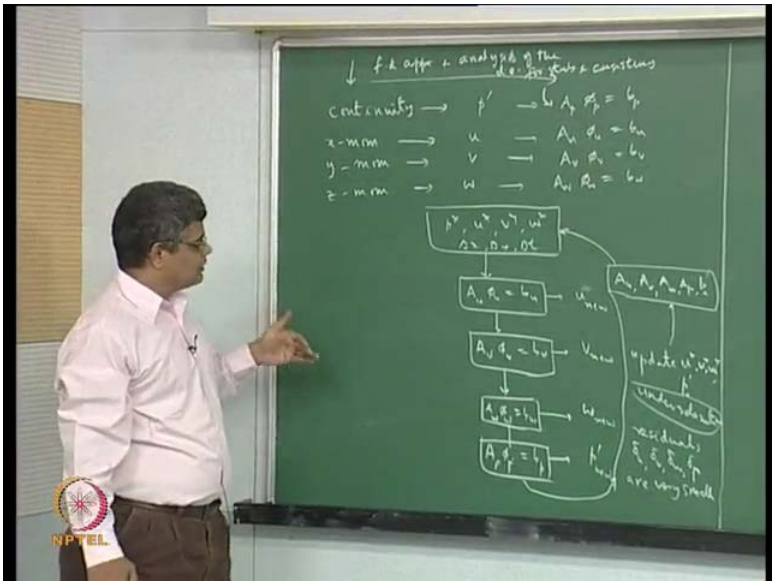
We start by having guess values, for example, p star, u star, v star, w star. So, these plus all the other information which is required for like, delta x, delta y, delta t, all that is known for us. And we use this to solve A u phi u equal to b u using an iterative method, for example, the gauss seidel method, for a certain number of iterations to reduce the residual to a reasonably low value; for example, by three decades, by three decimal places, something like that. So, we solve this to get u star or u new, at all the grid points. And after this, we solve A v phi v equal to b v; and then, from this, we get v new; then, we solve A w phi w equal to b w; from this, we get w new. And finally, we solve A p phi p prime equal to b p, from which we get b prime new.

We evaluate the residuals for each of these, and based on these things, we update v star w star and p prime star; and with these things, we reevaluate, A u, A v, A w, A p and again the b i's, that is, b u, b v, b w and b p, and from this, we go back. And we go

through this loop several times and we use under relaxation here. So, we under relax in evaluating from the new values - the updated values - here and then we go back, we go through this loop several times; and then, we monitor, we go until the residuals of each of these equations. So, we can call them as, delta u, delta v, delta w, delta p, become very small.

So, in this we way, we go through this loop several times, and it is for this reason that we have to solve these things several times like this. We make use of an iterative method, because these are evaluated with assumed values of these things; therefore, it is there, there is no point in evaluating this exactly right in the beginning itself. So, we go through this loop several times using an efficient solve, for a discretization scheme here, which is based on, which is known to give us astable and consistent solution.

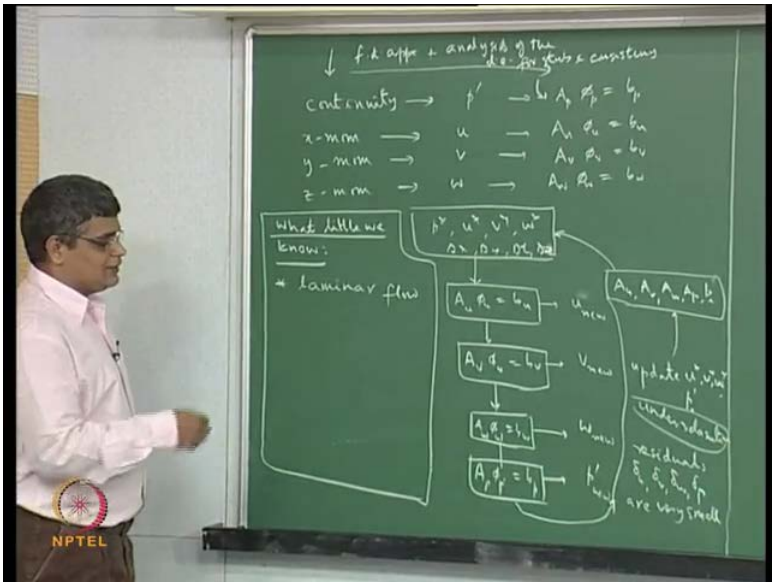
(Refer Slide Time: 12:11)



So, it is we first start with the equations and then we convert them into the discretized equations in the form of matrix, by substitution using finite difference approximations for all the derivatives, and any linearization that is needed for this plus analysis of the discretized equations for stability and consistency; to arrive finally at these equations, this form of this. So, this is the overall process for us to get a solution for a given problem. And we know all this process, we have looked at every stage of this process to say that, yes, we can go ahead and confidently do this. Although we can say this at this

stage, there is much more that needs to be done, because what we have done is subject to severe restrictions, which we can list down here.

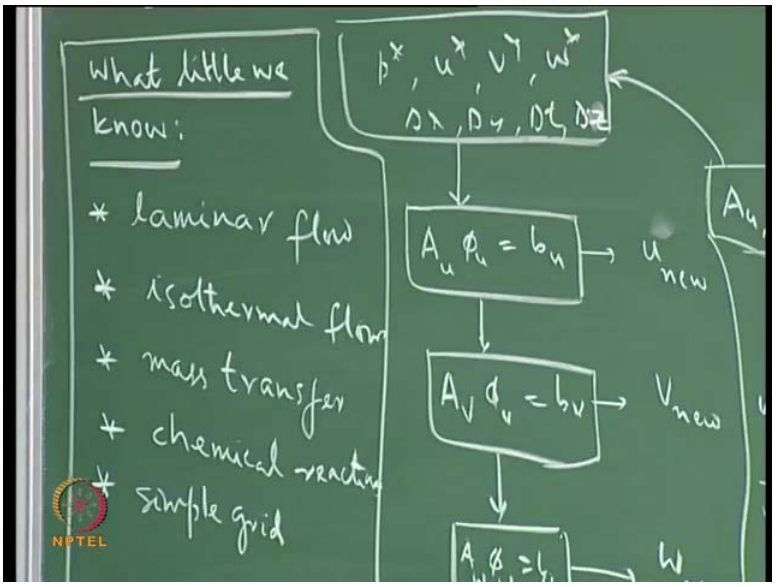
(Refer Slide Time: 14:13)



What little we know: the first thing that you should say is that, we have done equations only for lamina flow conditions, because we are solving the continuity and the momentum equations, which are specifically the lamina flow. And we all know that, when the Reynolds number increases, then the lamina flow solution is no longer applicable, why? Because if we are dealing with, for example, a steady flow condition, then we may not have delta t in our calculation method; we may have just the time independent form of equation, but turbulent flow is never steady and nor is it one-dimensional or two-dimensional. Here, we have put only the two-dimensional part of it; of course, since we have put w, also this is w, z delta z will also come into picture.

So, this calculation procedure is adaptable to steady flows, in which case we do not have to worry about the time term; and two-dimensional or one-dimensional in which case we can drop out, for example, delta z and delta y, but those kind of assumption of steady flow or one-dimensional flow or two-dimensional flow are not possible for turbulent flow. So, that is why this solution method is restricted only to lamina flow in the way that we see, we will see how we can do calculations for turbulent flow; so, but this is one of the limitations.

(Refer Slide Time: 16:12)

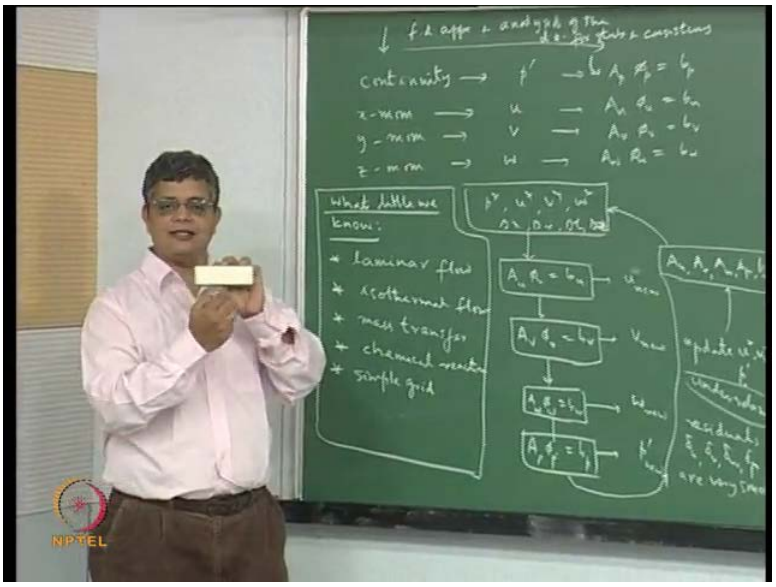


The second limitation that we have here is that, this is true only for isothermal flows, because we have not included in the calculation, the energy equation. So, that means, that we can only deal with isothermal flows. And there is no provision in what we have done so far for any mass transfer; for example, if you are looking at air conditioning type of application or a drying kind of process in a chemical or a process industry, there is definitely mass flow; if there is dissolution, there is absorption, all the cases there cannot be treated, because we do not have a mass transfer, we do not have the capability to deal with mass transfer in our governing equation and we have neglected chemical reactions.

If you are looking at some sort of chemical reaction, for example, some gasification or combustion or something like that, in such a case, these equations do not contain any such information. So, these are also practical cases, so which we can extend the equations; and right now, these kinds of problems cannot be tackled in the methods that we have done here.

Another severe restriction that we have is, that we have taken a simple grid, a simple grid which can be easily represented in x, y, z coordinates, where the overall flow domain - the boundaries of the flow domain, for example, this may be the flow domain, the boundaries of this flow domain here are represented as part of constant x line here and constant y line here.

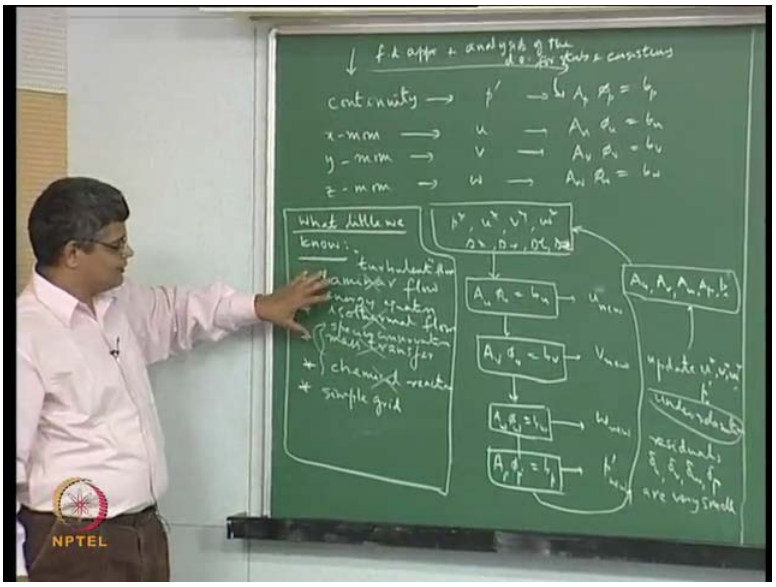
(Refer Slide Time: 18:19)



So, in that sense, this is restricted only to, for example, the rectangular kind of flow domains. If you have a flow domain in which this is there and inside that you have a chalk piece, and you want to find the flow over the chalk piece and look at what is the drag force acting on this; then, we cannot use this method, because we cannot represent this chalk piece in terms of simple Cartesian and coordinates. We have certain lines here, the surface here, which cannot represented as constant x or constant y or constant z, like that. So, in that sense, this is whatever method that we have used, that we have developed here is applicable only for the case of simple grid. Typically, when we are looking at a realistic flow conditions, then we have more than a simple grid, that is, as a requirement. So, in that sense, simple grid assumption is very severe assumption that we have.

So, with all these limitations, whatever that we have come up with so far is not really sufficient and it is not up to the mark, it does not make us any kind of specialist in the computational fluid dynamics; it does not allow us to treat meaningful cases - realistic cases - using the methods that we have developed so far. So, we need to overcome these limitations, in order to make our CFD more widely useful, and for us to gain some respect in terms of what kind of problems that we can do. So, this is where we would like to spend the next ten to fifteen lectures on tackling some of these problems.

(Refer Slide Time: 20:28)



We will consider the case of turbulent flow, and we see what do, we need to do in order to able to represent turbulent flow. We will see that, that turbulence is been is considered as one of the unsolved problems of fluid mechanics. These days where we have direct numerical simulation, we do not really know to what extend it is unsolved; we do not really know, whether we have solved the problem of turbulence or not; definitely it is one of the challenging problems and definitely it has lot of difficulties associated with that; and especially when you couple turbulent flow with chemical reactions, then it becomes really a humongous problem, which cannot be addressed in it is entirety in all it is complexities that are typically found in practical applications using CFD.

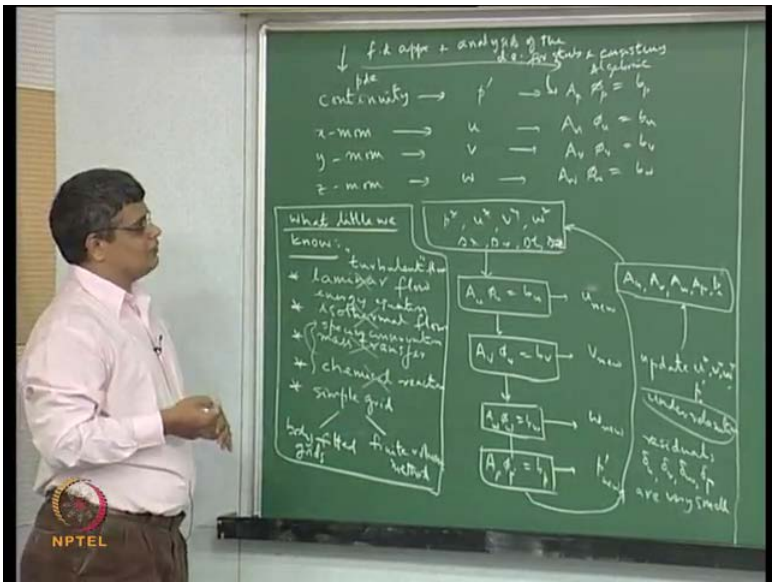
So, we need to make approximations and we need to bring in models. So, we need to see how we can deal with turbulent flow within the context of CFD. And we can easily get around this problem of isothermal flow by including the energy equation; and the mass transfer problem can also be readily overcome by including a species conservation equation; therefore, one can consider two constituents of the fluid medium, and then, we can see how they can go into each of other and so on. So, this is again we need to have extra equations for this; chemical reactions is again something that can be readily encountered, readily accommodated, through the species conservation equation, in which we can have a species generation term due to chemical reaction. So, we can take these things together and then extend this. So, these extensions are necessary for us to say that,

and they are not only necessary, but these are in a way fairly straightforward in terms of incorporating them into the overall calculation method.

We will see that each of these equations, that are, that are necessary tackle this real fluid situations can be put in the form of the typical scalar transport equation; and therefore, it can be readily converted into some A phi equal to b representing that particular, for example, the species conservation equation or the energy equation. And therefore, it can be readily incorporated into this; we may have another A s phi s equal to b s, where s is a species conservation equation.

So, it is because of the facility with which we can tackle all these extra things, within this overall CFD solution method, that we need to understand how it can be done and then readily incorporate this. Now, when we want to buy overcome this simple grid assumption, it becomes a truly complicated problem; it is not something that, that can be done readily, and in fact, that development of the generation of non-orthogonal grids or body fitted grids in the early eighties, actually was considered one of the great one of the significant advances in the development of CFD, and it has really brought CFD into the regular day to day engineering calculations. And we need to do a significantly more to overcome the simple grid assumption, and what we do is, we look at the basic ideas of how we can do this; and we consider two approaches, one is where we transform the computation from the x, y, z, into an arbitrary coordinate system, and this coordinate system is designed in such a way, that it can tackle the real world complexity through the use of body fitted grids.

(Refer Slide Time: 25:10)

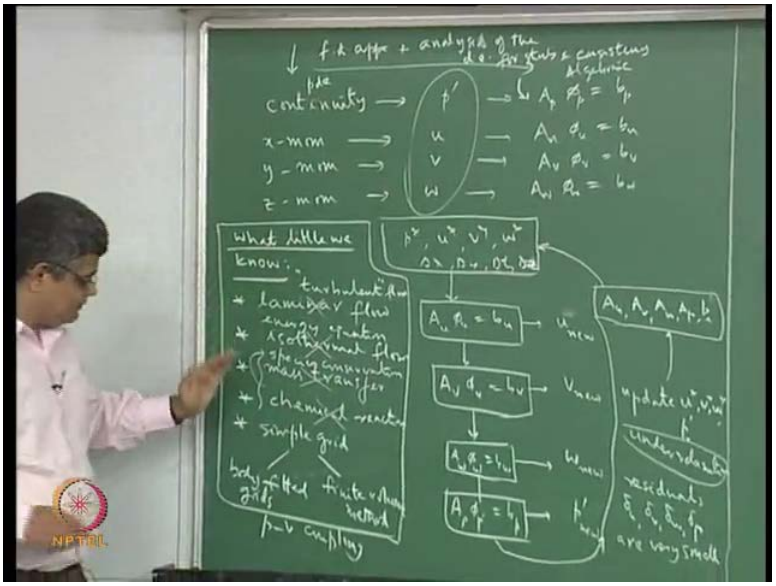


So, we no longer do the calculations in x, y, z, but in some other coordinates system and that coordinate system is such that, in that coordinate system, we can we can have some grid lines which wrap around this chalk piece within this flow domain. There are some other coordinate lines, which wrap around this geometry; so, in that way we can we can do that. So, that is a generic method which can using, which we can tackle complex geometry. And the other is the finite volume method, which fundamentally changes the way that we can go from the equations - from the partial differential equations - into the discretized equations ultimately. So, both this approach and this approach affect the way that we go from the partial differential equation into the algebraic equations.

So, when we want to overcome the simple grid limitation, we have to make a significant effort to one of the major part of the CFD approach; that is, to go from that partial differential equation into algebraic equations, but once we come here, this procedure is more or less same, but some aspects of this calculation method will now be different in these two cases. For example, this is what is known as structured grid, and this is what is known as an unstructured grid. And the way that we solve an a phi equal to b type of thing for a structured grid, can be different from the way that we solve for an unstructured grid. In some ways, not only is this changed, but also the calculation of these things may also changed. And the other part that is changed in this is, we go from the staggered grid approach to a collocated grid approach, because when we deal with

complicated geometries, we do not want to deal with four different grids for the four different variables and that may bring in special problems.

(Refer Slide Time: 27:55)



So, in such a case, we have to look at the coupling of these four equations. So, that coupling is also something that we have to look at.

Pressure velocity coupling, for that we have adopted for the simple grid will now have to be reexamined and then re -tuned, so that it can work on these complicated grids. So, these are the aspects that we are going to look at in the next several lectures. We will start with the simple things first, because these affect only the equations, the rest of the thing does not change. And then, we will come to the grid part, where we want to tackle complex geometry and we look at each of these issues.