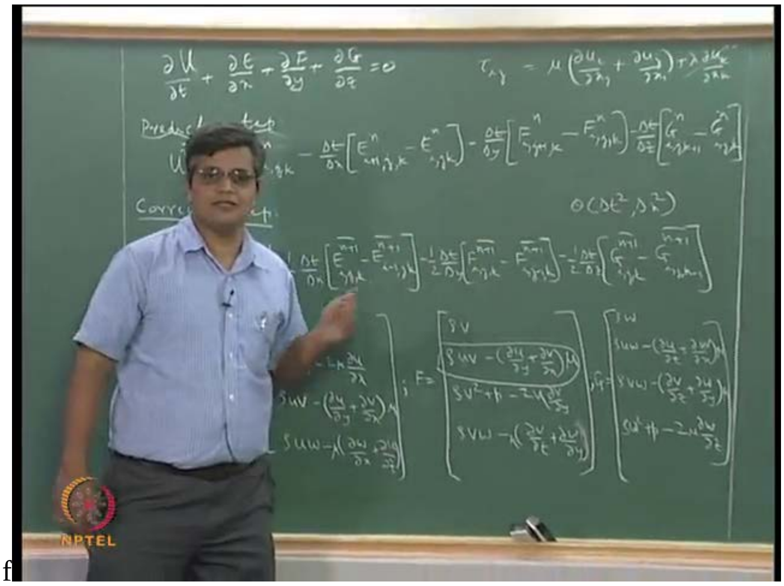
**Computational Fluid Dynamics**
**Prof. Sreenivas Jayanti**
**Department of Chemical Engineering**
**Indian Institute of Technology, Madras**

**Module No. # 04**
**Solution of Navier-Stokes equation**
**Lecture No. # 18**
**Topics**
**Stability limits of MacCormack scheme**
**Limitations in extending compressible flow scheme to incompressible flows**
**Difficulty of evaluation of pressure in incompressible flows and listing of various approaches**

(Refer Slide Time: 00:12)



f

So, now we have a method, the MacCormack method for the solution for a Navier-Stokes equation. Can it be applied for all cases? Is this a panacea? Is this a method that we can use to solve all the equations?

Let us examine the issue. This we have seen is the predictor, corrector step of second order accuracy, both in time and space. So, that is very desirable aspects and it is also both the steps are explicit n plus1 bar is given only in terms of n here and n plus1 here is given only in terms n plus 1 bar. We have a sequential evaluation of of all the variables, so this looks like a good way of solving the equation and it has proved to be successful, but like many explicit methods this has stability requirements.

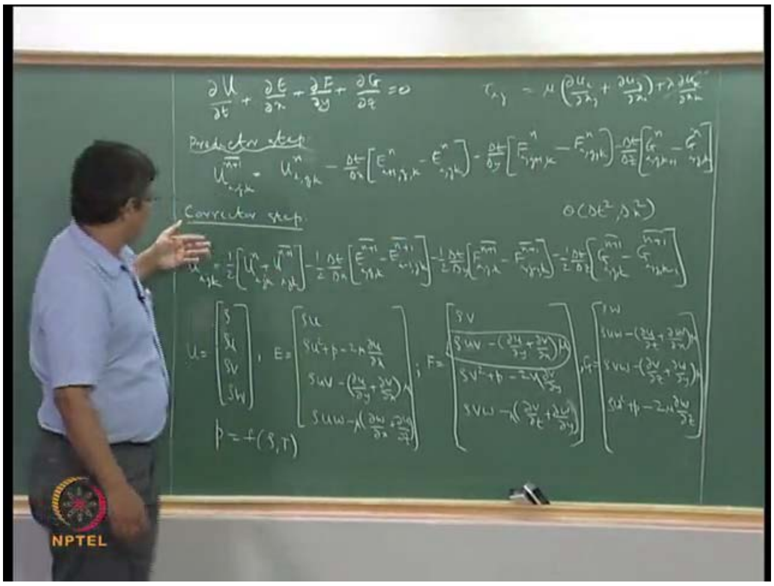
You cannot have an arbitrary delta t and delta x. Here, we have delta t, delta x, delta y, delta z and we known that this equation, although we have put it like this, has an advective component. We also have a diffusive components coming from the sheer stresses and we also have the source term coming from the prissier gradient ok.

You have a pressure gradient here, and here, and here. Fortunately for us, the pressure gradient is a not a functional the variable that we are solving in a direct way. We can see that pressure is influenced by this, and pressure is also given by the density, so in that sense it is in it is non-linear. The source term is non-linear, but it is not directly coupled to the evaluation of rho u rho v rho w in a such a way, it that it appears as variable in terms of rho u, v, a, w here.

So, in that sense although the pressure is a non-linear variable in terms of density to the equational state for that particular fluid, as per the equations are concerned; here it is not appearing as source term, where it is a function of phi ok.

So, if you write this as a generic scale transport equation, then the source term as a function of the scalar variable phi is not very clear in this. So, we can only say, it is indirectly implicitly coupled into this. So, we have a source term which is independent of phi here and we have an advective term and diffusive term.

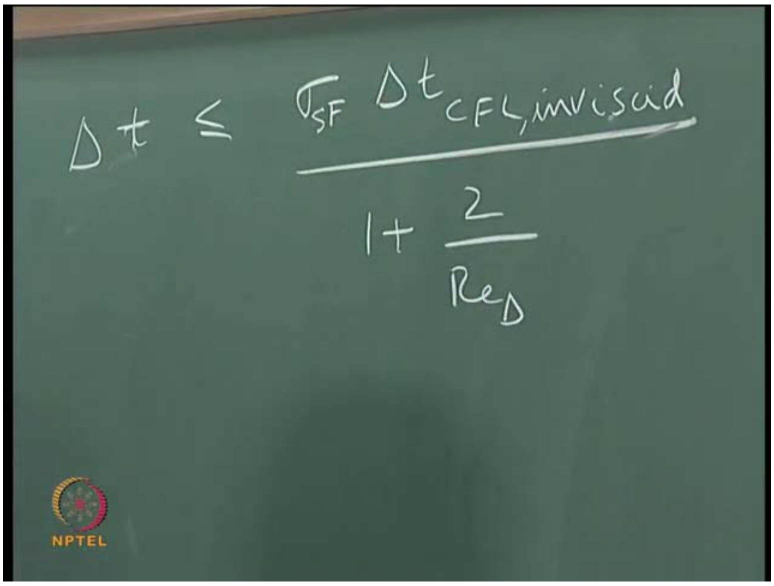
(Refer Slide Time: 00:12)



So, we can expect something like courant number kind of limitation coming from the advection term and we have, for the diffusive term we have the diffusive terms delta t by delta x where that kind of combination of limit that should (()). We not only have those two, we also have the multi-dimensionality term, we x direction y direction and z direction.

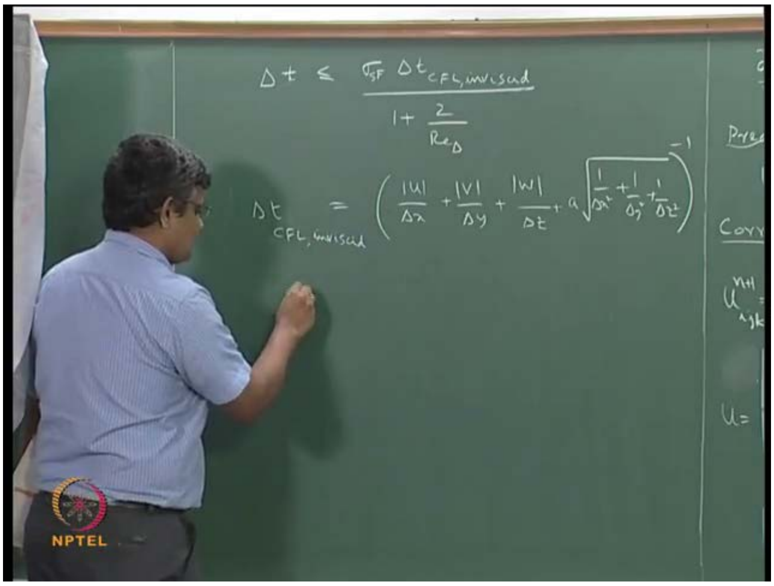
So, we can expect further degradation of the stability limits, especially for the (()). So, what the overall stability condition on this of this method? It is not easy to evaluate and as we go to more and more complicated things, the stability condition probably can be evaluated either heuristically or empirically. It can only be estimated, may not be possible to [cough] evaluate this especially or through a close form type of solution given especially. The fact, that pressure one of the variables that is coming here is coming through indirectly through an algebraic equation of state, the form of which is not readily evaluated. So, from this one can only in a semi empirically l way maybe, we can write down the stability condition.

(Refer Slide Time: 04:50)



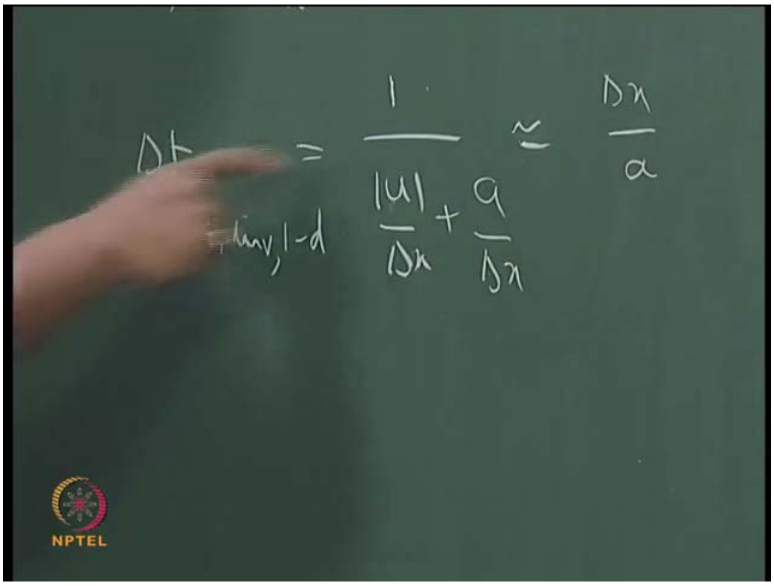

The stability condition for this is expressed in the form of delta t. Delta time should be less than delta t CFL, so that is a courant friedrichs limit. This is in the case of inviscid, let me put this as times; a factor of safety sigma. Let us put this as sigma s F divided by a function, which is based on Reynold’s number and it is given, it estimated to be 1+2 divided by Re delta. So, we have the overall the stability that is maximum delta t, we can have here is; obviously, very closely linked to the inviscid flow in which case the viscous stresses do not come in to the picture becomes hyperbolic function.

(Refer Slide Time: 05:53)




This in viscid CFL condition for the delta t can be written as the delta t CFL inviscid meaning; obviously, viscosity is is zero. Given that, we have three-dimensional flow with not necessarily zero u v w components, it is plus a times square root of 1 by delta x square plus 1 by delta y square plus 1 by delta z square and this whole thing inverse ok. So, one can say that, if it were one dimensional flow, so that v zero, w zero, and delta y and delta z do not come into the picture.

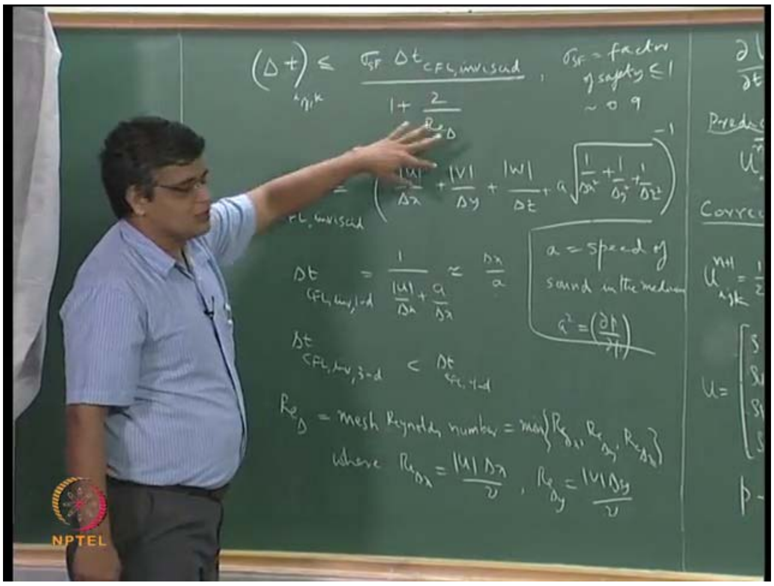
(Refer Slide Time: 07:04)



This delta t CFL inviscid becomes 1 by u by delta x plus a by delta x ok. So, if u is very small compared to this, then this becomes delta x y a, so this is, in the case of one dimensional thing. This is just to illustrate the form of the this delta t ok. So, the delta t is delta x divided by a; where “a” is the speed of sound and is given such that, a square is equal to d rho by d t under is oentropic conditions. So, a in that sense this is related to the equation of the state and so, this is property of the medium itself speed of sound in the medium. If it is gas, gas its medium and mixture of gases then the corresponding things.

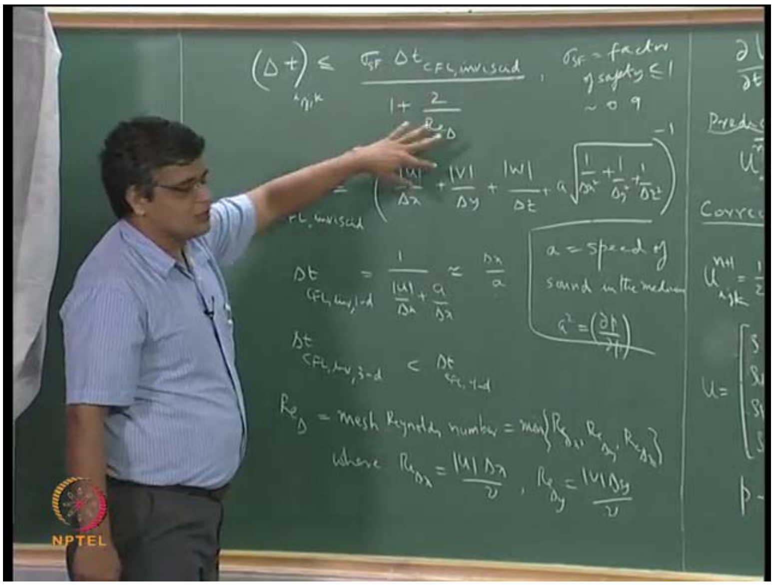
So, what we see from this? The inviscid flow delta t is now related to the speed of sound and it is also related to the velocity of the flow. In three-dimensions, it is not only this, but you have a contribution coming from v and w and we also have these things.

(Refer Slide Time: 08:12)




So, you can say that delta t CFL inviscid 3d is less than delta t CFL 1d ok. Now, these CFL is what is coming here in the 3d and this is being multiplied by a factor of safety. Sigma SF is the factor of safety because we cannot evaluate this exactly and this is less than 1 for it, to be a factor of safety and is typically taken to be about 0.9 ok.

(Refer Slide Time: 08:12)

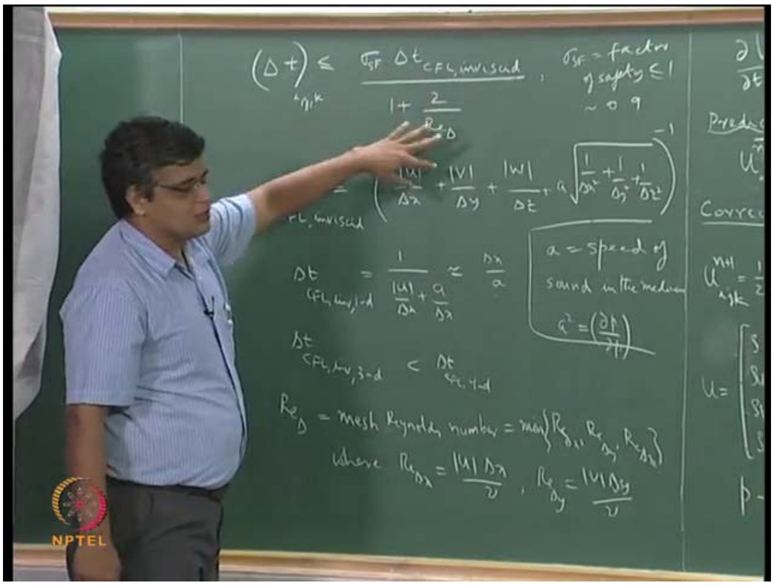



So, that means that the delta t, that we can use for these is less than what we can use in one the dimensional limit and it is less than that because you are using factors of safety. This whole thing is less than the one dimensional limit because you have a viscous effect coming through the Reynolds number and the Reynolds number delta here we called a the Mesh Reynolds number. This the minimum of Re delta x, Re delta y, Re delta z because now, we have three different velocities and three different have spacing, where Re delta x is defined as u delta x by nu; where nu is the kinematic viscosity. Similarly, Re delta y is obviously determined by v delta y by nu and so on ok.

So, you you evaluate. So, if you want to find out what is the minimum delta t that I can use, then it is highly dependent d on the local values of u and the local value of delta t that you can have. So, you you evaluate that particular point, so we are looking at delta t at a particular grid point ijk, so you evaluate uijk delta x and then vijk delta x and wijk delta x.

So, based on these, you evaluate and you take the smallest of these as the Mesh Reynolds numbers and you put 2 by the smallest of this. That gives you the largest possible of the contribution from the Reynolds number that is added to the 1, so this obviously, greater than 1. So, the delta t that you can actually use is now evaluated based on this formula. For each of the points throughout the domain, you have for all ijks and you take the smallest of those values in order to ensure stability.

(Refer Slide Time: 08:12)



Some people may say, you take the largest value because it is an estimate and for example, in case u is 0, if one of the velocity component is 0, then this becomes very small; this becomes very large and delta t becomes almost 0 ok. So, to avoid those kind of things, you can take a representative values of this but the point remains that the delta t CFL that we have for a pure convective thing, which is based on the delta t maximum is delta x by the wave velocity.

In this particular case, the delta x divided by the speed of sound and we know, the speed of sound is typically high so; that means, that delta t is small and the actual value is made smaller. By the fact, you have three-dimensions, so the CFL value itself is smaller and you have an addition contribution coming from the Reynolds number; that is a viscous stress component and there is also an addition factor of safety which is coming less than these.

So, because of this the delta t ijk is less than the ideal value for the one dimensional case and we have to choose the smallest of I, the delta t for all the grid e points to progress further in this.

So, based on this consideration, we choose the delta t and once you fix the delta t, then we substitute this into each of these. Then we go from n to n+1 at the end of n+1, we revaluate all the factors contributing to the delta t and then we pick a new value of delta t, then we go in and proceed further. So, as we go from n to n+1 to n+2 to n+3 so on, the delta t value may change. It may become lesser or it may become greater depending on how the velocities are changing and the how they are feeding into the inviscid limit. Also, the into the reynolds number here to get the overall viscous limit of cf, the delta the stability condition, which is an estimated quantity ok.

So, this method is a workable method and with this stability condition given like this, it is a completely workable method. In the sense, that we can go ahead and solve this with some initial conditions, then, go on up to the to investigate the evolution of the flow variables as time progresses. Now, can we extend this to compares incompressible flows because we have said that, that is the main area of interest, especially in in chemical and process industries were typically; we are looking at Mach numbers of less than point 0.3.

So, in which case you have very little compressible flow effect, so in such a case what happens to this? What happens especially to the delta t? We can have the method here, but in order to implement this we need to make sure that delta t is smaller than whatever the limit says.

Now, as we as flow becomes more and more incompressible that is d rho by [fl] yes, this is the speed of sound, so as the u has the flows becomes more and more incompressible, the speed of sound in that medium; it increases more and more for a purely incompressible flow. The speed of sound is infinity ok.

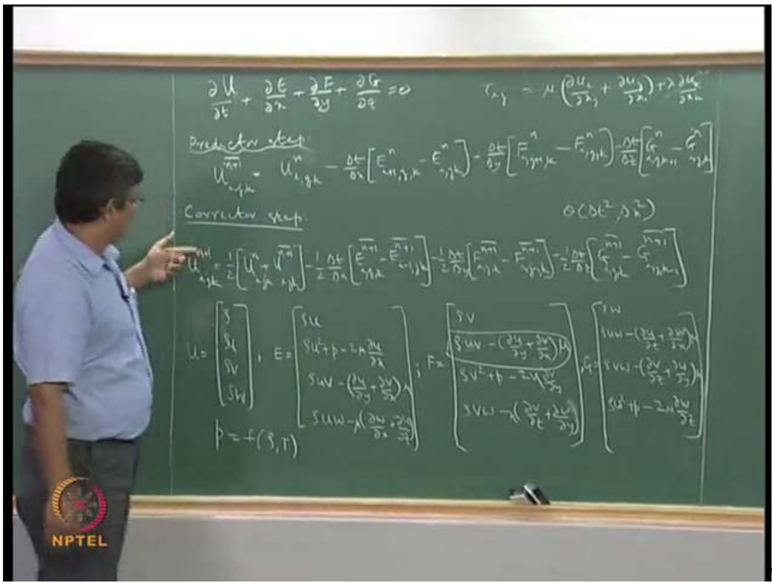
Now, what happens when the speed of sound increases? As it increases, this increases, so one by this we have put here here one by this increases, so as a increases; this becomes small and this becomes large. So, the effectively the delta t inviscid limit is given by delta x by a and as a increases; delta t inviscid decreases. So, as a increases, delta t cf l decreases.

So, and as the flow becomes more incompressible, then the delta t that you can employee for this method decreases and you can take smaller and smaller delta ts only. In the incompressible limit a is infinity so; that means, this method would not work ok. So, the explicit method, which typically has ah a courant number type of constant and in the case of viscous flows, we have additional constant. So, it cannot be directly applied to incompressible flows because of the stability requirement.

Now, if we cannot apply the explicit method, can we go to implicit method? We because we know that implicit method many times have unconditional stability. In fact, t there is an implicit form of the MacCormack method itself and there are many other methods, even explicit and implicit all of them. All the explicit methods have typically, some sort of constant, especially when you dealing with a set of equations in three-dimensions. Like, what we have here and when you to go implicit method for the given scalar transport equation with constant coefficients? You can have unconditional stability, but if you have to large a time step, if you take delta t as being large for example, ten times the inviscid limit that you get from these things, then the error introduced in the time evolution will will become large because any scheme that you have here like this has for example, this has a second order ah error with respect to time.

So, if as delta t becomes large, the error in the evolution of evaluation of the time variation, the time dependent term will become large. The problem that we have with the coupled equations with the Navier-Stokes equation, that if you want to evaluate for example, the u velocity, not only do you have u, but do you also have rho which is indirect p, which is indirectly coupled to u and then you also have v here and then w here.

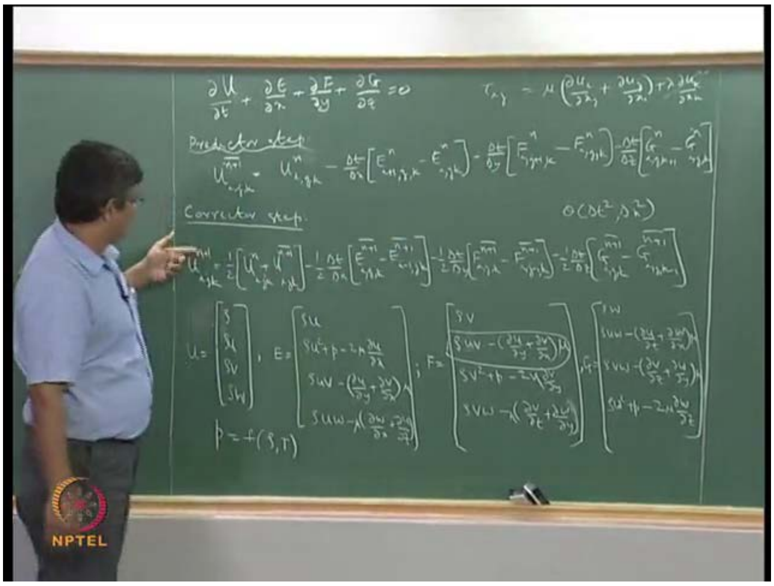
(Refer Slide Time: 00:12)



0

So; that means in order to evaluate u, you need to know all the other variables that are incorporated in this. Now, if you want to solve only the u momentum equation, then your implicit method may give you unconditional stability, but in order to do that you also need to simultaneously solve the v equation, w equation. If you make large errors in the evaluation of u because you have used large time steps and therefore, you have misrepresented this, then that plays havoc with the values of v and w which are integral part of this. When you go to the evaluation of v and w equations, you are carrying forward a large time error and in the evaluation of u and therefore, since the evaluation of u also effects the evaluation of v and w, the coupled nature does not allow you to have two large time steps. So, you can typically have only may be 5 times or 10 times [ah] the inviscid limit for ah [it] explicit limit for for an implicit calculation and that arises primarily because of the inter-coupling of the Navier-Stokes equations.

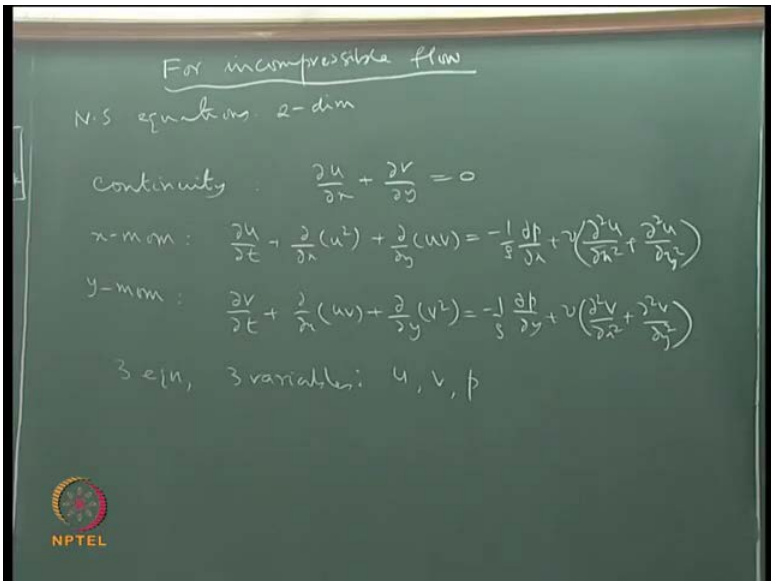
(Refer Slide Time: 00:12)



So, direct applications of these kind of methods, which have an ah time step limitations arising from compressibility considerations like this speed of some limit that we have, will not allow us to extend these things to directly to incompressible flow. So, although this McCormack method is quiet useful and it has been applied for the solution of the coupled equations, it it is not readily extendable to full fully incompressible for calculations because the corresponding time step that is required for this will become ah impractical, so we need to find new methods, new ways of doing this.

So, that is where the straight forward extension of methods developed for compressible flow to incompressible calculations typically fails and and we would need to look at the special difficulties that are associated with the incompressible flow of calculations to arrive at methods, which are tailored to those kind of flows k. So, let us just write down what the equations of incompressible flow are and then, we can understand what the difficulty that is there is.

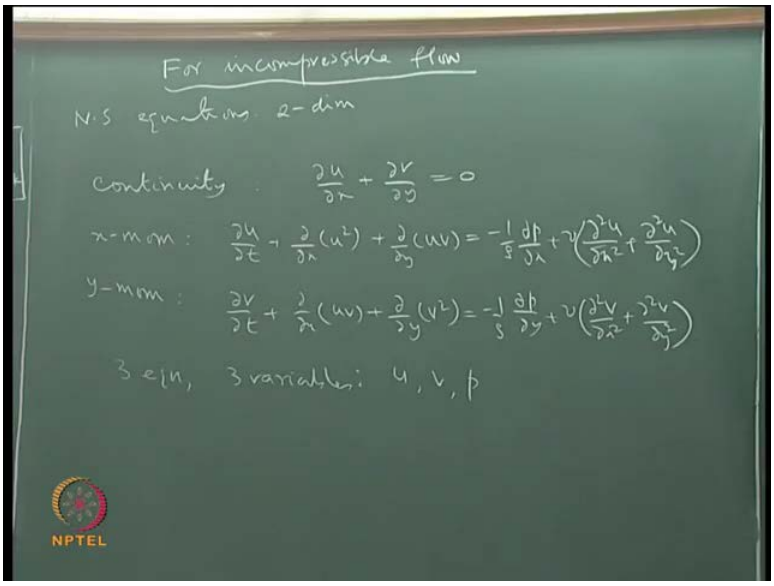
(Refer Slide Time: 21:58)




We know that for incompressible, the Navier-Stokes equations become like the continuity equation. Let us say that we are dealing with two-dimensional flow, so as to make the equations a bit smaller. The continuity equation is dou u by dou x plus dou v by dou y equal to 0 and the x momentum equation is dou u by dou t plus dou by dou x of u square plus dou by dou y of u v equal to minus 1 by rho dou p by dou x plus u times dou square u by dou x square plus dou square u by dou y square, assuming constant property, so that mu the viscosity the kinematic viscosity is also constant, rho is obviously constant because the considering the flow to be incompressible. We can write down the y momentum equation as dou v by dou t plus dou by dou x of u v plus dou by dou y of v square is equal to minus 1 by rho dou p by dou y plus u times dou square v by dou x square plus dou square v by dou y square.

So, we have three equations and we have three variables and whatever variables we have, we have uv and p, so ah from a point of view of of of the closedness of the problem. We have as many equations as a vary number of variables, so that in principle, we do not need any other information except of course, the initial and boundary conditions as per requirement as per the nature of the problem and so on k.

(Refer Slide Time: 21:58)




So, in principle we should be able to calculate the get a complete solution of u, v, p variation with x, y and time by solving these equations with appropriate initial and boundary conditions. Now, when we want to implement this in our ah standard cfd form, we can we can write the corresponding thing as as scalar transport equation like this, this is a standard form that we have in two dimensions, we can put these equations in this form k ok.

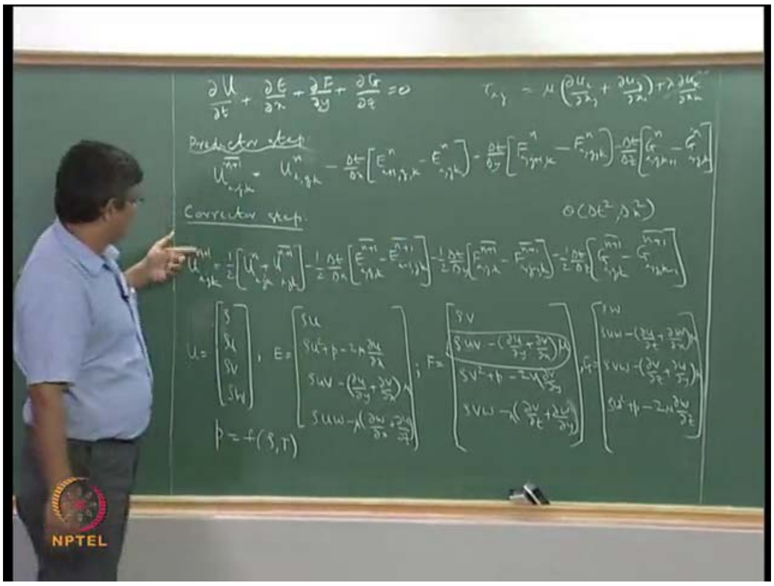
Now, where phi becomes either u, v and a thing like that. Now, the difficult thing that we see here is that now, the x momentum is a statement of the conservation of u, ah u the velocity compound to the x direction; therefore, we expect that this equation should be solved in order to get u ok. So, the value of u is more or less determined by the momentum balance because u represents the momentum for unit ah mass in that particular direction. Similarly, one can say that the value of v that is the vertical velocity component, which is which is also the specific y, momentum is given is determined by the y momentum balance, so we have a natural equation for the evaluation of u and v ok.

So, out of the three variables, we have the x momentum equation to get this and y momentum equation to get this. So, the remaining variable is p pressure and the remaining equation is the continuity equation.

So, this means that we have to get p from the continuity equation and the difficulty is that there is no p term. This pressure is not appearing in the continuity equation here, so when we consider incompressive flow equations, if we make use of the u momentum equation x momentum equation for u and y momentum equation for v, then we have the only other equation; the continuity equation left to evaluate pressure. Unfortunately, pressure does not appear in the equation at all so, that means that u cannot get pressure by solving this equation.

We do not have the problem in compressible flow because in compressible flow, we are not solving directly for pressure. We are solving the continuity equation for rho and the x momentum equation for u, y momentum equation for v and z momentum equation for w ok. So, we have we have making use of the natural equations here and rho appears in the continuity equation because we have d rho by dt plus all those things, so the continuity equations, which is used to evaluate the fourth variable which is rho has the corresponding variable appearing as the main ah ah term ok.

(Refer Slide Time: 00:12)




So, the continuity equations for a compressible flow of is d rho by d t plus d by d x j of rho u j equal to 0. So, from that point of view, we can therefore write this is equal to minus of this and this can be used to evaluate rho ijkn plus 1 from rho in and so on k.

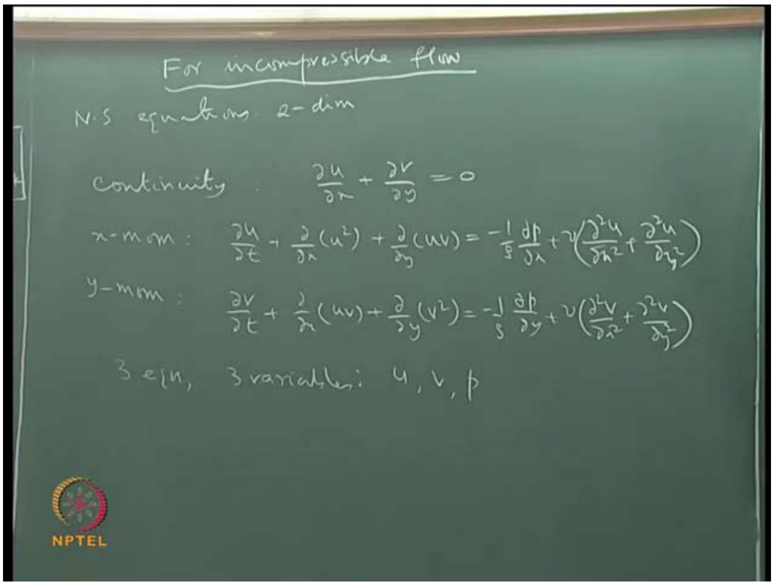
So, in that sense for compressible flow, we have no such problem because we have we have the fourth equation is used to evaluate the fourth variable. The fourth equation does have the fourth variable as the primary variable in this as the main variable in this and we evaluate the pressures from an algebraic equation of state from known density and known temperature. So, the difficulty of evaluation of pressure does not arise in compressible flow, but it arises in incompressible flow because of this specific problem k.

So, question is how to evaluate pressure? So and we know that ah the speed of sound is linked to the variation of density with pressure, but for an incompressible flow that linkage between density and pressure is not there. It is a no matter what the pressure is, the density does not change. So as far as an incompressible flow equation is concerned, density and viscosity are imposed variables. They have no relation to the ah flow variables which are u, v, p and ah u, v, w and p.

So, even if pressure changes, the density do not change in incompressible flow. So, in a way incompressible flow has no sense of absolute value of pressure; it is only the pressure difference that is the pressure gradient that comes in to the picture in incompressible flow.

So, that that marks ah a major difference between compressible flow and incompressible flow that is the feedback effect of the pressure on the density and through the density onto the equations is non non-existent in in ah in incompressible flows. It it is the relative pressure difference that affects the velocities and of course, the velocities have to change in such a way that they also obey the continuity equation.

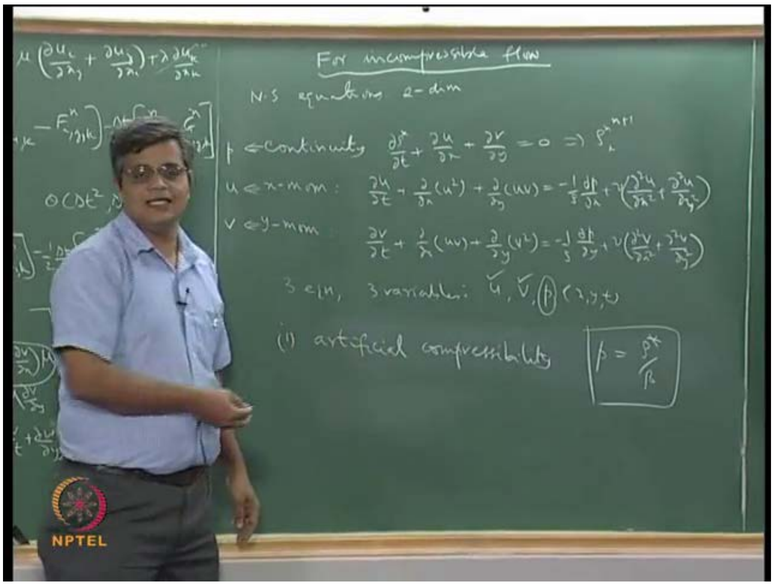
(Refer Slide Time: 21:58)




So, we have only an indirect linkage between the pressure and the continuity equations. That linkage has to be somehow brought in to play, when you want to solve these three equations together for this k. One can say, why not make use of the continuity equation for u and this equation for p and this equation for v? That kind of thing is not not possible. It is like trying to control a tiger by holding its tail, k then ah you know that we cannot control the tiger just by holding the tail because it will probably *pounce* on you and then eat you up, but if you were hold it by the neck and then ah do something there, then you can definitely control it ok. So, by using the continuity equation here and then the pressure here, will lead to that kind of situation because of the strong inter coupling of ah of the three equations. It is not possible to do in other way than to use the primary equations for the primary variables ok.

So, the most logical thing is is to solve the x momentum equation for u and y momentum equation for v because u and v represent the specific momentum in the two respective directions. We expect that momentum balance will give you the u and unfortunately, we cannot solve these equations unless specially specified. We can get the pressure only from the continuity equation, but the pressure is not there as a variable in that. So, how to get around the difficulty? There are several methods that have been developed, specifically for ah to get around this and one of these things is to introduce artificial compressibility ok.

(Refer Slide Time: 32:23)

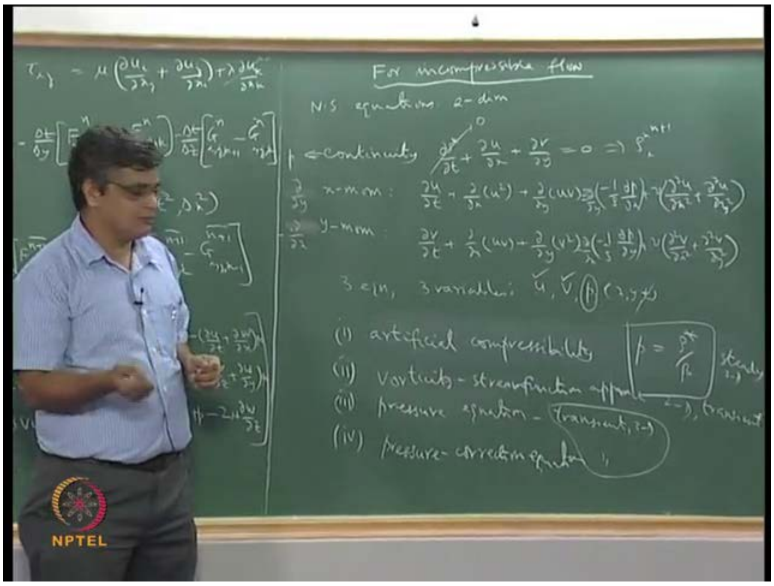



So, we say that since, we are not able to get ah pressure from the continuity equation if we introduce an artificial compressibility. For example, we say that pressure is given by rho by say beta and where beta represents the compressibility, so that if you now introduce the term like let put this as rho star, so you introduce d rho star by d t in to the continuity equation and you solve this for rho star i n plus 1. Once, you get the rho star in plus 1, you can go back to this equation and get the corresponding p here k.

So, in that way, we can introduce an artificial compressibility and add a term to the continuity equation, which allows you to evaluate this d rho by d t as a function of the changes in velocities. Therefore, you are thereby, you are coupling the equations, coupling the pressure and the continuity equation with the momentum equations through this artificial compressibility. Now, this is only a fictitious thing ok.

This is something that we are adding. It is not the true density; it is not the true coupling because there is no coupling between pressure and density for incompressible flows. So, the resulting solution is not correct for all trouble that we take. We get a variation of u, v, p with respect to time and also x, y, but we know that is not correct because we are solving the wrong equation.

(Refer Slide Time: 37:05)

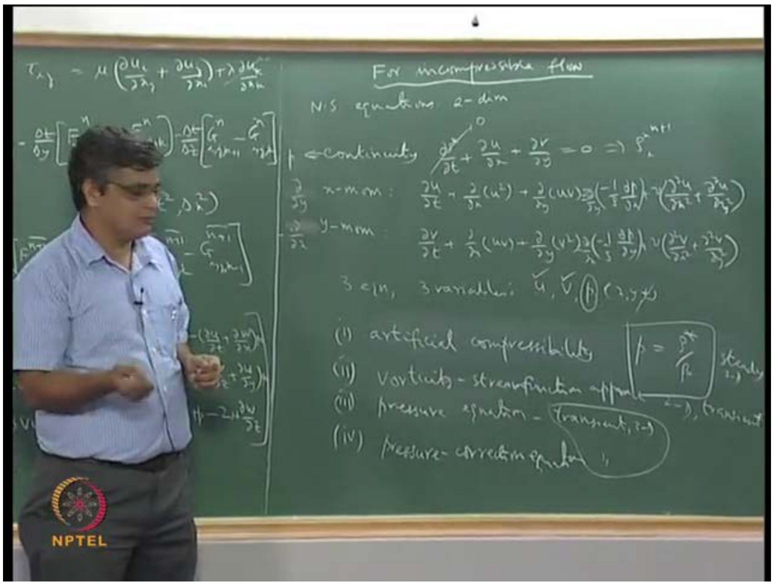



This term should not be there, but the steady solution that we have will obviously, have the contribution coming from this to 0. So, under steady conditions, this term will go to 0 and the steady velocity feel ah that we get u, v, p, x, y, k where, the time is not there is such that it will satisfy all the three equations, k the three equations, the three equations without this (())

So, the artificial compressible method is a way of getting the correct solution, which is valid only for steady flows, but ah it would not work for unsteady flows, but by introducing an artificial compressibility. You can create an artificial linkage between a fictitious density and pressure and we can make use of an extended form of the continuity equation to solve. For that, artificial density term and then bring into that to create thereby, a bridge between the pressure and the velocity variables to the continuity equation.

So, this method will work and we will look at ah this method in detail later. It has been successfully applied for many steady flow solutions.

(Refer Slide Time: 37:05)

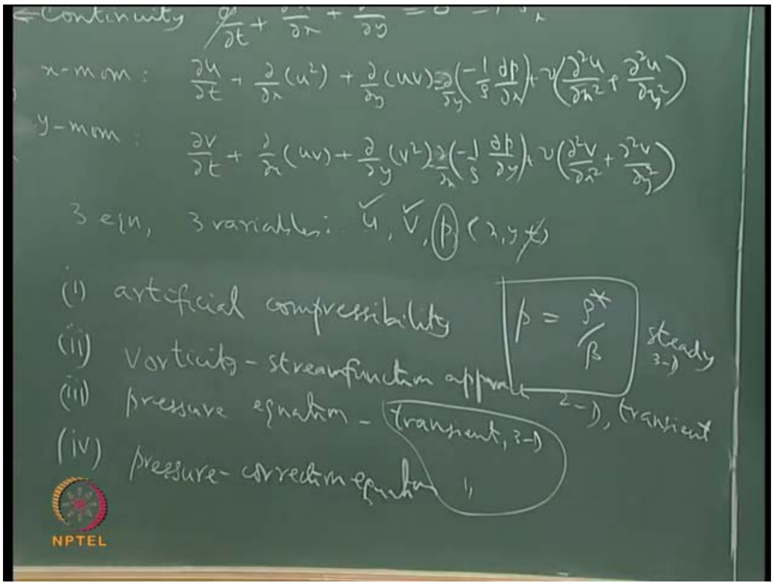



There is another possibility, since pressure is the headache variable which is causing us all the headache. Why not just eliminate it from the Navier-Stokes equations? If we eliminate it, then we can ah come up with a set of equations, which do not have pressure; therefore, we do not have to solve it. We, if we pressure does not appear in any of the equations, then we do not have to solve it.

Is it possible to eliminate it? Is possible to eliminate pressure for example, if you take the derivative with respect to y of every term in the x momentum equation and you take derivative with respective x of every term in the y momentum equation and then you subtract this from this, then k there will be changes here.

This term here will have dou by dou y of this and this term will have dou by dou x of this. Now, when p is continuous and rho of course is constant, then this term will become minus dou square p by dou x dou y and this also minus dou square p by dou x and dou y. When you subtract the two, these two will cancel out and the resulting equation that is the equation resulting from the subtraction of the dou by dou x of the y momentum equation from the dou by dou y of the x momentum equation, we will only have velocity variables with respective x y and t and they will not have any pressure.

(Refer Slide Time: 42:06)

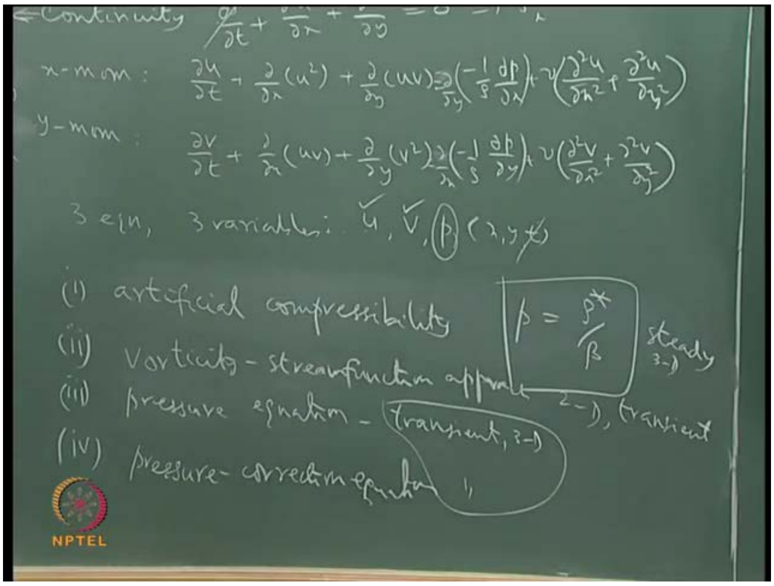



We will show that we can define a new variable called vorticity and using that, we can get a vorticity transport equation, which looks very similar to the generic scalar transport equation for which we we have come up with ah discretization methods ok. So, we solve that vorticity transport equation.

For the vorticity, which is the combination of the velocity gradients and we can also introduce the concept of stream function and using the stream function, we can eliminate this. We can come up with a new set of variables vorticity and stream function approach in which the velocities do not appear at all and the pressure do not appear in the combined vorticity and stream function relations.

So, the three equations that are appearing here can be clubbed into two equations, which are partial differential equations ah which can be solved together vorticity and stream functions. From these things, we can get velocities and velocity u, v components and we can solve an additional equation for pressure that is called a Poisson equation pressure, which can also be derived from these things ok.

(Refer Slide Time: 42:06)



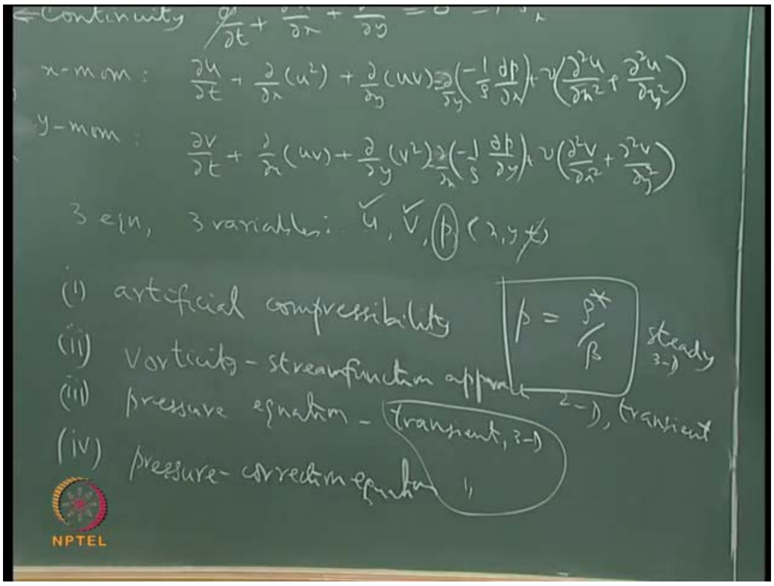

So, we we evaluate u, v, p that is the velocity components and ah and pressure. From these equations, in a roundabout way we transform these three equations into two coupled equations involving vorticity and stream function we solved and using standard methods because these from these equations look very similar to our generic scalar transport equations.

We solve them using the standard methods and which will give us a stream function vorticity from which, we can deduce the values of u and v and once deduce the values of u and v, we bring them back here. Then, we do further manipulations of this and we derive a Poisson equation of a pressure, which has u, v as the variables and the velocity gradients has a variables. Since, we already know the velocity gradients of ah since we already know velocities u and v, we can evaluate the gradients and there we can convert this into Poisson equation of a pressure and we solve that equation to get pressure.

So, the computation of u, v, p is now broken in into two steps, may be three steps. One is solution of vorticity and stream function together in the form of two coupled partial differential equations from that deducing u and v and from this solving ah another partial differential equation for pressure.

So, that kind of approach is also a very widely used, but this equation is valid only for when you can define a stream function. Stream function is defined only for two-dimensional flows so, this method is applicable only for two-dimensional flows, and this method is applicable only for steady flows ok.

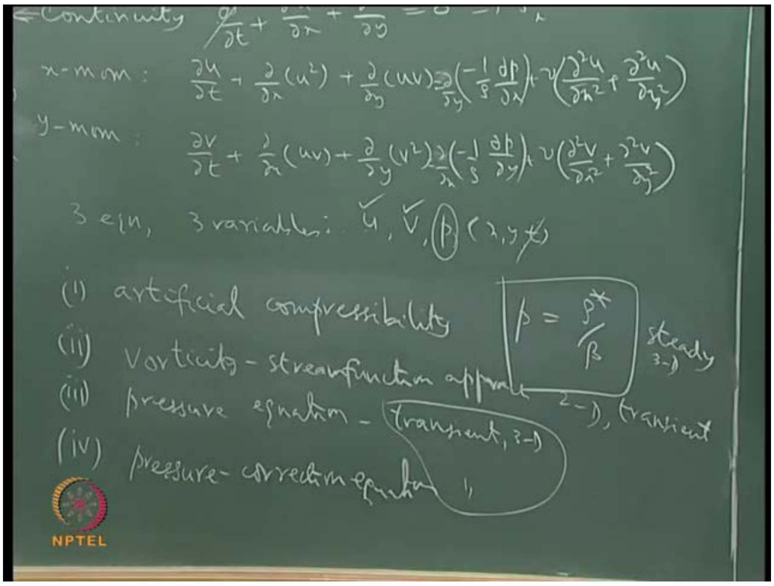
(Refer Slide Time: 42:06)




So, this is applied applicable for steady three-dimensional flows. This is applicable for two-dimensional transient flows. In that sense, both the methods cannot be used for the generic case of transient three-dimensional flow ok.

So, now if you want a transient three-dimensional flow, then we have to do something more and ah there again we make use of ah further manipulations of these things to get that equation pressure equation, which we already obtained as part of the vorticity of stream function method. So, we solve for the momentum equation and pressure equation in a coupled way k and ah so, that is called a pressure equation approach. This is applicable for transient 3d and the fourth method, which is much more often used, especially for incompressible flows and especially for ah a chemical and process engineering application is instead of solving for pressure equations pressure directly, we solve for a pressure correction ok.

(Refer Slide Time: 42:06)




So, we solve, we derive a pressure correction equations by re, by manipulating these equations and then we solve for the pressure corrections. We can come up with an overall solution approach, which is specifically tallied for incompressible flows.

So, all this ah four methods and many variants and other methods are derived specifically for incompressible flows to get around the problem that is unique to incompressible flows. In the sense, that pressure is not coming as a as a direct variable in any of the major equations. There is no equation for pressure and this so, we have to reformulate in a certain way, in a different way, so as to recover pressure from the continuity equation ok.

So, that is how we we can ah evaluate the flow variables for incompressible flows. The last two methods are applicable for the general case whereas, these two methods are specifically for either steady flows or 2d flows. The artificial compressibility method is can be is such that we can directly apply the methods developed for compressible flows, like what we have seen the McCormack method either in its explicit form or implicit form can be used for artificial compressibility thing. The other type of ah methods also can be used for vorticity stream function approach, but this is only for a two-dimensional case so and these are the special methods. The specific aspects of the solution are still common to all those things because all the four methods will involve either the generic scalar transport equation, which is a time dependent ah term advection diffusion and a source term or in these three cases in addition to that at least one scalar transport equation you also have an additional either

Laplace equation or Poisson equation that has to be solved. Laplace equation or Poisson equation is a special form of the scalar transport equation, for the case where you have ah for the r steady case k steady non convective case or steady diffusive case is is Poisson equation when you have (())steps k.

So, in that sense the methods we have developed for the general scalar transport equation are still applicable and they can still be used here, but we have to come up with a rephrasing of the equations of the governing equations, in order to able to solve them for incompressible flow. So, in the next ah lectures, we will look at these methods and see how these can be implemented.