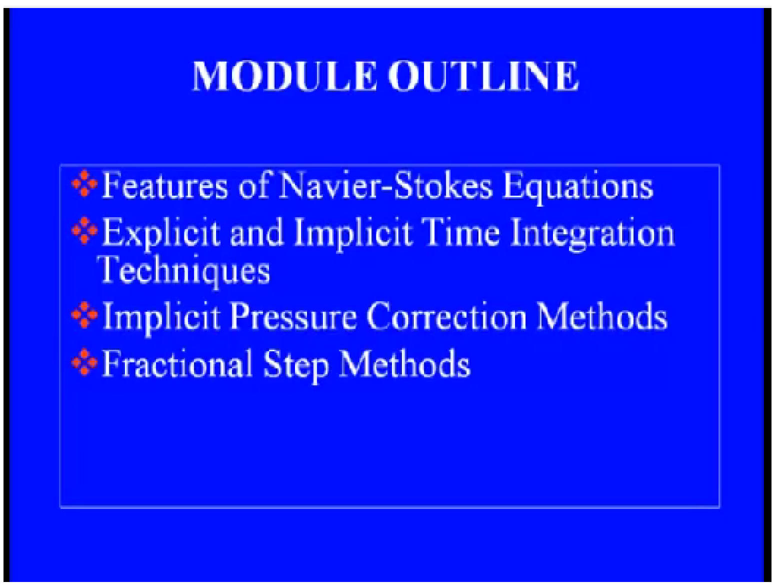
Computational Fluid Dynamics
Dr. Krishna M. Singh
Department of Mechanical and Industrial Engineering
Indian Institute of Technology - Roorkee

Lecture – 35
Special Features of Navier-Stokes Equations

Welcome to module 8 on Numerical Solution of Navier-Stokes equations. So this module represents the heart of a CFD, that is to say the application of the techniques which we have learnt so for, their application to solution of real life flow problems. So in this module, we will have a brief look at the fundamental features of Navier-Stokes Equations, what differentiates them from the generic transport equations which we have solved earlier using finite differences, finite volumes and finite element methods.

(Refer Slide Time: 01:07)



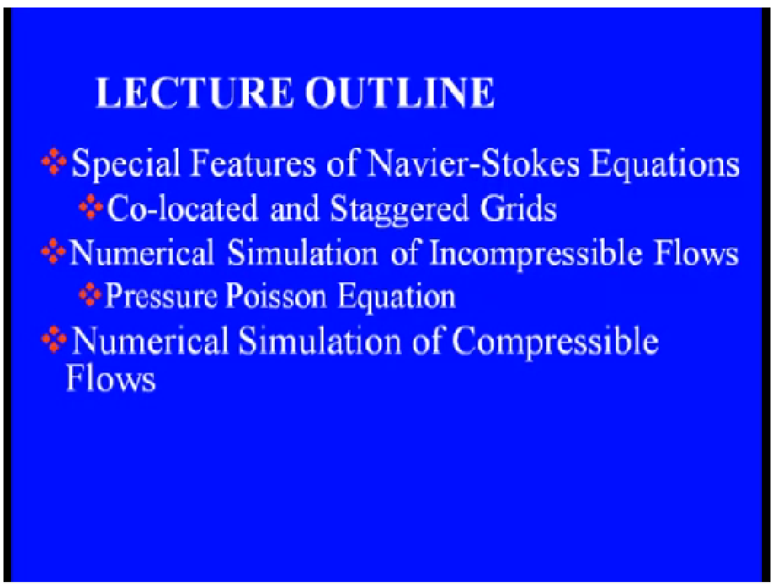
And then we will have a look at Explicit and Implicit Time Integration Techniques as they are applied to the solution of incompressible Navier-Stokes equations and we would look at Implicit Pressure Correction Methods which are popular for the solution of steady state problems and we will also look at Fractional Step Methods.

Now this module would make use of all the knowledge which you have gained earlier on discretization schemes, that is finite difference, finite volume, or finite element method, time integration techniques and the taking for solution of algebraic equations. So we will not go into details of the spatial discretization using a particular technique or a particular time integration scheme or use of specific algebraic equation solver.

We would focus on the fundamental algorithms and fundamental features which we require in the solution of Navier-Stokes equations. So we will presume that the user is familiar with a particular discretization scheme, finite element, finite volume or finite differences, that is why this has been made and then which are the explicit and implicit time integration schemes which one can use and what are additional things which are required in the solution of Navier-Stokes equations.

Now in the first lecture, we would focus on the special features of Navier-Stokes equations, what differentiate these equations from the solution of ordinary single variable transport equations.

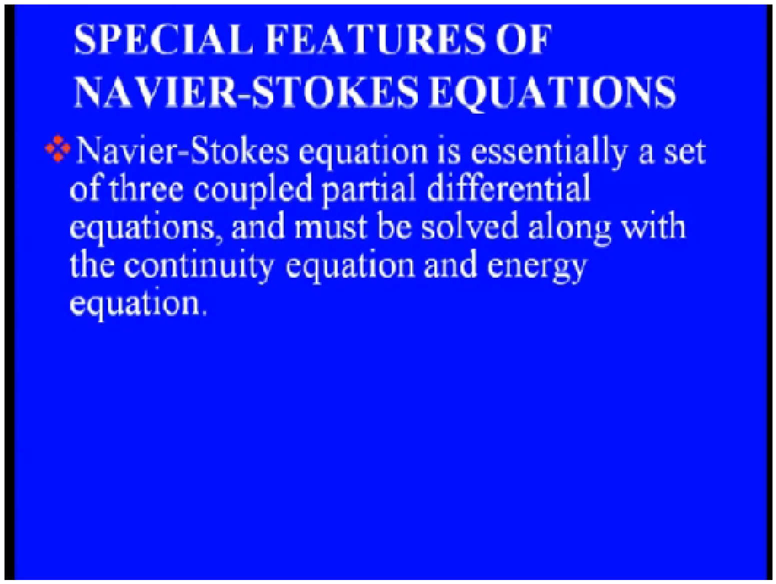
(Refer Slide Time: 02:48)



So what are the special features that would form the crux of the today's lecture. We will look at the different types of grid arrangements which we can use in Navier-Stokes equations which is facilitated by the nature of the equations. In particular, we will have a look at what is called co-located grid arrangement and what is meant by standard grid arrangement. In particular context of the structured finite difference or finite volume discretization schemes.

Then we will have brief look at the Numerical Simulation of Incompressible Flows, in particular, we will derive one equation which is essential in the solution of incompressible equations and it does not come directly from our conservational laws which we call Pressure Poisson Equation and we will have brief look at the Numerical Simulation of Compressible Flows just in a generic fashion.

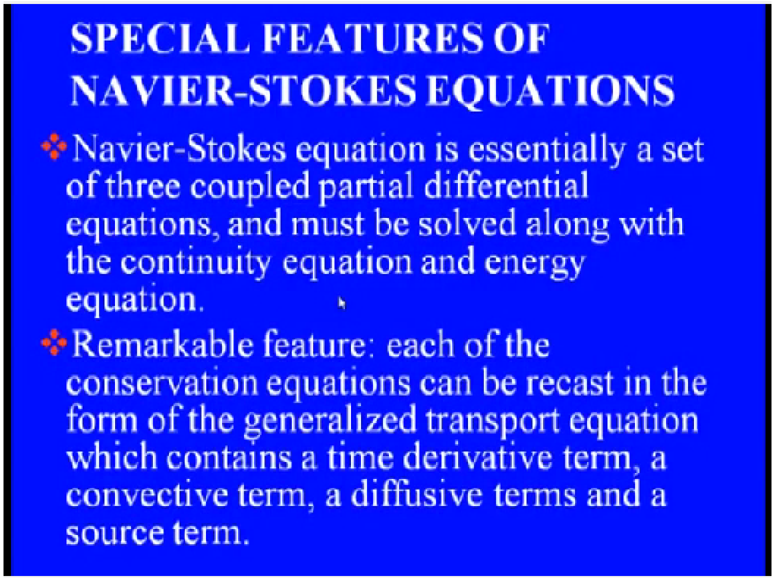
(Refer Slide Time: 03:46)



Now what are the special features of Navier-Stokes equations. When we looked at Navier-Stokes equations, what we call them essentially our momentum equations for a Newtonian fluid and momentum equation is a vector equation so that essentially means that Navier-Stokes equations are a set of 3 coupled partial differential equations. It is not a single equation in contrast to a scalar transport equation and in addition, it must be solved along with the continuity equation and energy equation.

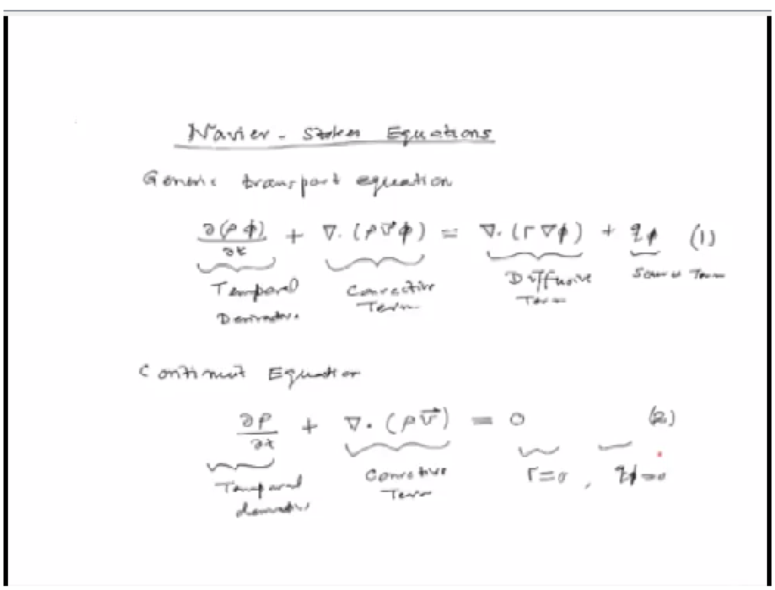
So this is what differentiates the Navier-Stokes solution from the solution of a transport equation or let us say simple heat conduction problem. In fact, in few of the literatures, it is very often would when we talk about Navier-Stokes equations, what we mean is, we mean the correction of continuity, momentum and energy equations taken together because all of these must be solved, all of these are coupled and they must be solved together to obtain our flow variables that is to say the velocity components, density, pressure and temperature.

(Refer Slide Time: 05:03)



Now another remarkable feature which we have we can observe is, that each conservation equation can be recast in the form of generalised transport equation which contains a time derivative term, a convective term, a diffusive term and a source term. To clarify this point, let us have just brief look at 2 of the equations, the continuity equation and the momentum equation in the vector form and let us see whether our contention holds good or can be recognised the similarities between Navier-Stokes equations system and our generic transport equation.

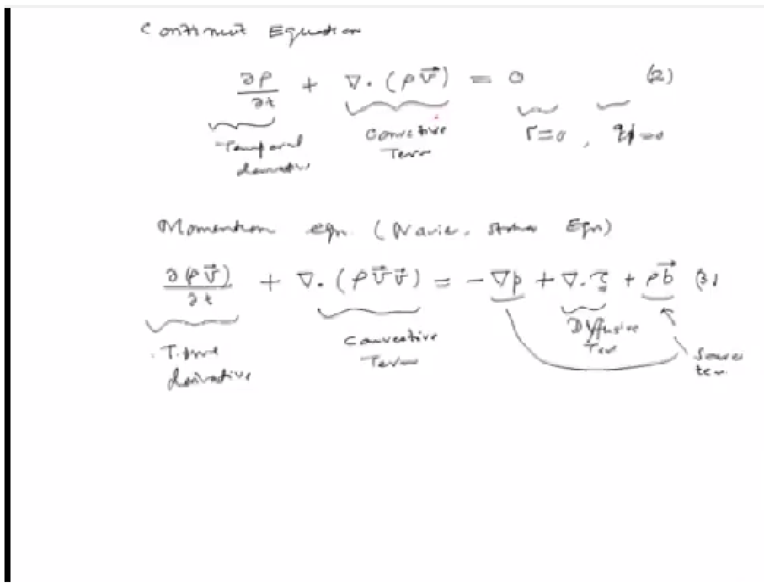
(Refer Slide Time: 05:36)



So first let us write our generic transport equation, del of rho phi/del t+divergence of rho v phi=divergence of gamma*gradient of phi+we had a source term for q phi. So the first term, this was our temporal derivative. This is what we called the convective term, this was our diffusive term and the last one was our source term. Now let us have a look at continuity equation, del rho/del t+divergence of rho v=0.

Now if you compare the 2 equations, this generic scalar transport equation and our continuity equation, what we can observe is simply if we put phi=1 and gamma and q phi as 0, we get our continuity equation. So the continuity equation also has a temporal derivative term and a convective term. We can assume here that gamma=0 and q phi=0 but in form, it is very similar to our generic transport equation.

(Refer Slide Time: 08:11)

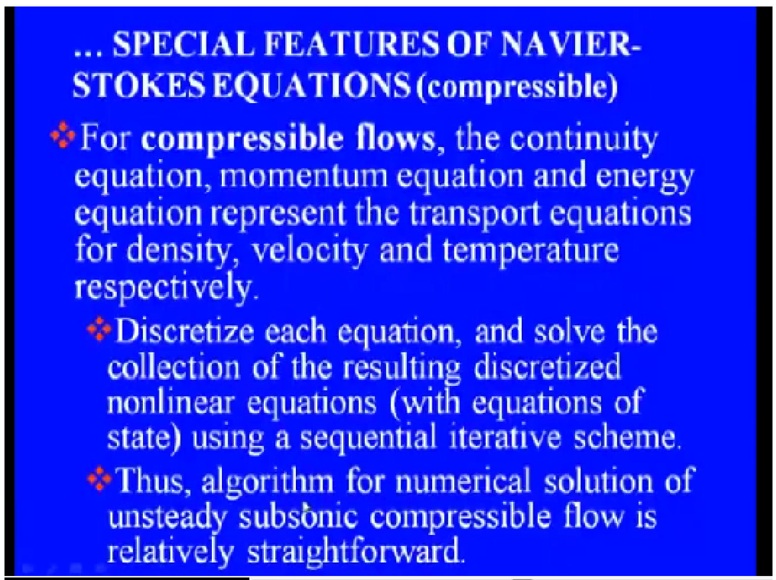


Similarly, let us write our momentum equation which we also called as Navier-Stokes equations, Del rho v/del t+divergence of rho vv-gradient of p+divergence of tau+rho*b. So once again if you compare this equation with our generic transport equation, the similarities are obvious. We have got a time derivative term, a convective term, (()) (09:18) represent our diffusive term and this body force could be taken, body force along the pressure gradient, these 2 can be combined together in the form what we call the source term.

Okay, so from these continuity equation and momentum equation, it is very similar to that of our generic transport equation and that would allow us to make use of this test which you have learned earlier for numerical solution of generalised or generic transport equation. So that is why we said that each of the conservation equation can be recast in the form of generalised transport equation which contains a time derivative, a convective term, a diffusive term and a source term.

So all that it means is that we can chose a specific technique, let us say for special discretization and for time integration which we have earlier learnt for the solution of scalar transport equation and that scheme can also be extended rather easily, full solution of full set of Navier-Stokes equations.

(Refer Slide Time: 10:37)



Now if we deal with compressible flows, in the compressible flow of continuity equation, it is essentially a transport equation for density, the del rho/del t that is time derivative for density and plus divergence of rho*v that gives us convection term for the density momentum equation that represents a transport equation for velocity and energy equation represents the transport equation for either total energy or we can also transform it into the transport equation for temperature.

So in fact in the case of compressible flows, it is rather easy for us to combine these set of equations as continuity equation, momentum equation and energy equation in form of a generic vector equation representing transport of a vector quantity and apply a solution technique to solve this set of equations.

So all that we need to do in this case is, discretize each equation, solve the collection of resulting discretized equations which are non-linear and we have to supplement our equations, though by equations of a state for instance we have pressure coming into the momentum equation, we have got viscosity, we have got thermal conductivity coming energy equation. So we need to supply additional equations for the evaluation of these, equation of state is required for computational appraisal.

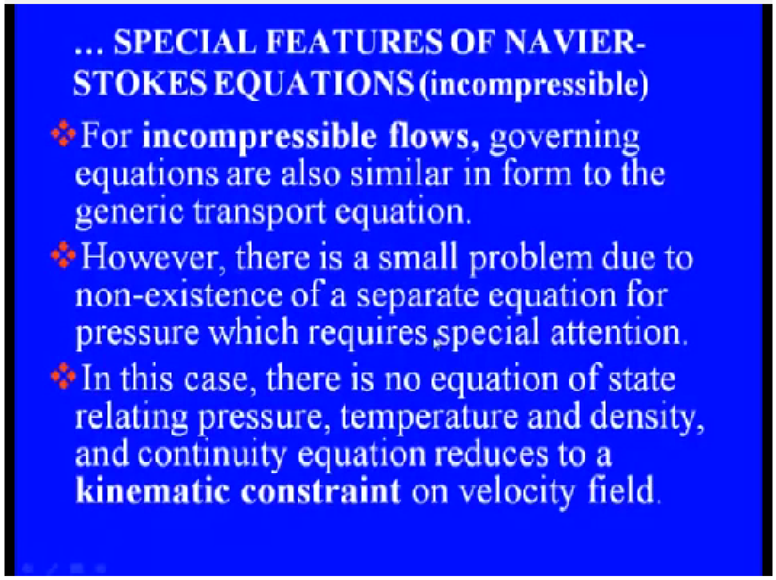
We also need equations which will give us temperature dependent viscosity and the thermal conductivity to enable us to solve the compressible flow problem. But as far as the conceptual solution process is concerned that is very simple that since each equation represents a transport in particular quantity, continuity equation represents transport equation for density.

X momentum equation is transport equation for let us say rho u, y momentum equation gives us transport equation for rho v, z momentum equation will give us transport equation for rho w and energy equation will give us the transport equation for let say total energy, discretize each one of them separately using same set of discretizing procedures and time integration schemes and solve them sequentially.

In fact, if you are using an explicit solver, the solution becomes very simple, we do not have to use a sequential iteration scheme in that case. So thus in algorithmic terms, the algorithm for numerical solution of unsteady compressible flows is relatively straightforward. Of course, there are issues which we will have to take care of when we go to a hypersonic compressible flows, there would be formation of shockwaves which have to be accounted for. So the solution process would be a bit involved, we have to take extra care in that case.

We are not going to discuss those issues in this introductory course, but nevertheless if you can choose fairly small time step and a good quality, a fairly refined mesh, we can solve our compressible Navier-Stokes equations for any mock number without much fuss. Of course that will involve huge amount of investment in terms of the computational time. Next, let us have a look at incompressible flows.

(Refer Slide Time: 14:26)



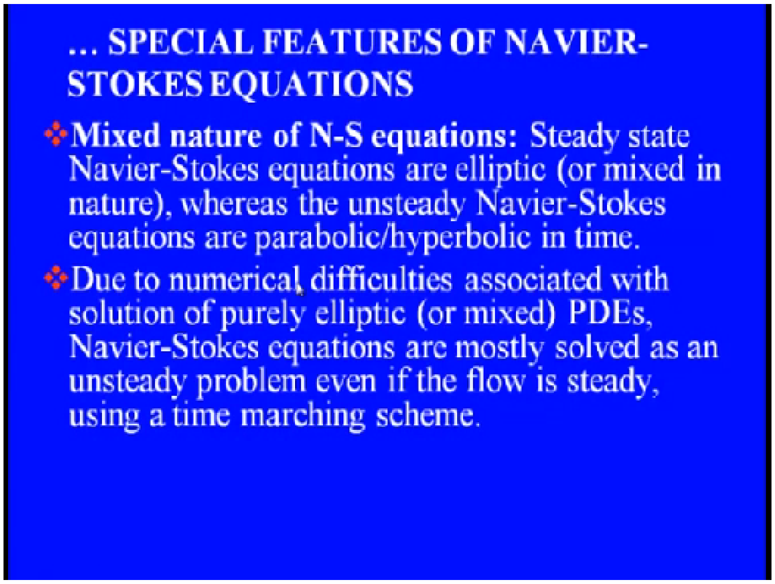
For incompressible flows again, the governing equations are in the form similar to the generic transport equation with one difference here, that in continuity equation what happens is, it is not a dominant equation in terms of density, density here is constant that we know. So the continuity equation cannot be said to be a transport equation for density. Similarly, there is no equation of a state.

Since we do not have a equation of state like p=rho rt for an ideal gas involved in compressible flow simulations, then how would we get a pressure and we do need gradient of pressure in a momentum equation. For this particular feature, it is worth requires special attention for the solution of incompressible flows and that would be one of the issues which we are going to address today.

Again please remember that in the case of incompressible Navier-Stokes equations, there is no equation of state which relates to pressure, temperature and density and in this case, continuity equation reduces to what we call a kinematic constraint on velocity field. We will have to say that divergences of b is 0.

So velocity field which we get as a part of a solution process, would be such that is divergences free which is essentially a kinematic constraint, it is not a dynamic constraint, but nevertheless for the satisfaction of continuity, this kinematic constraint must be satisfied by the velocity field which we compute numerically. So there are 2 issues that how do we get pressure and how do we satisfy this kinematic constraint imposed by the continuity equation. We would address specifically today.

(Refer Slide Time: 16:24)



Now what other features. If you can go back to the mathematical classification which we dealt of conservation equation which we dealt in module 2. We said look, Navier-Stokes equations, the non-linear coupled equations and they have got what we call a mixed nature given incompressible Navier-Stokes equation if you have say steady state then we say it is elliptic equation, if it were unsteady, we say it is parabolic.

When we go for compressible flows, the situation becomes even more complicated, that classification of the equation would depend on the basis of local mark number of the flow. So even if you are dealing with a steady state problem, there could be regions of flow where the equation is elliptic in nature, there are other regions where it becomes hyperbolic. So the equation is mixed and coming up with a single numerical simulation algorithm for a mixed equation is rather difficult.

We also saw one feature that if you look at unsteady Navier-Stokes equations, situation is slightly different, specifically with respect to time. Unsteady Navier-Stokes equations for incompressible flow, they are parabolic in nature. Similarly, unsteady Navier-Stokes equations would become hyperbolic for the case of high-speed compressible flows.

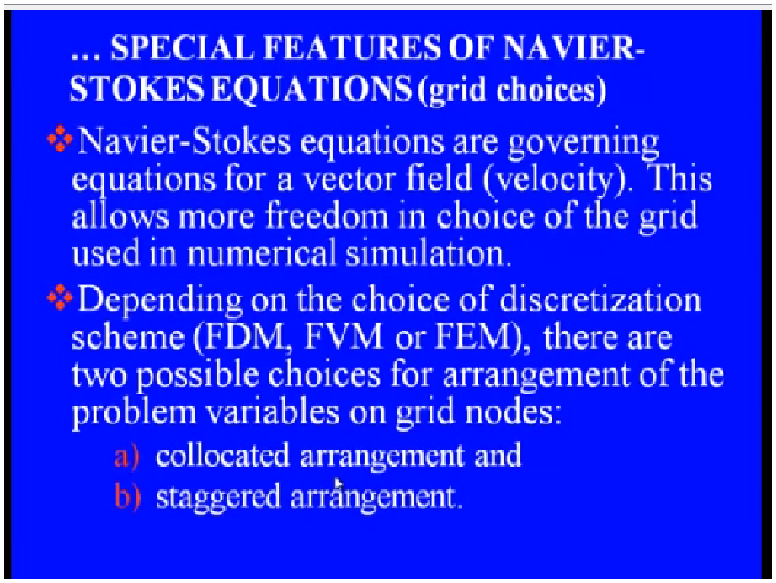
So this is something which we can utilise or we can exploit in the numerical solution because here now the nature is much simpler, it is not a mixed nature, so as a parabolic or hyperbolic in time for incompressible or compressible flows respectively. So can we make use of this particular thing.

So that is why we have this, there are numerical difficulties which are associated with the solution of purely elliptic in the case of steady state incompressible flows or mixed PDEs in the case of compressible flows or compressible Navier-Stokes equations. We rarely attempt to solve the steady state Navier-Stokes equations in either case.

In fact, Navier-Stokes equations are mostly solved as unsteady problem even if the flow is steady using a time marching scheme and what we say that we have to integrate in time for fairly long time instance. So longtime solution is what would be or this longtime solution of this transient Navier-Stokes problem would (()) (19:06) solution of actual steady state.

And this methodology is universally used in the solution of Navier-Stokes equations whether we are dealing with incompressible flows or compressible flows that we would always solve a steady-state flow problem as an unsteady problem. If your steady state solution is our only aim, we would normally use implicit time integration scheme which will allow us to use large time steps to reach the steady state very quickly.

(Refer Slide Time: 19:46)

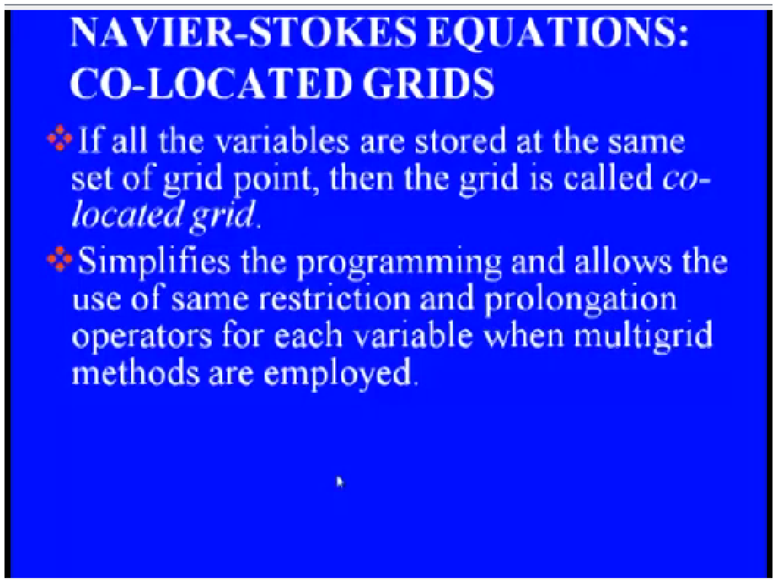


Now Navier-Stokes equations, they are governing equations for a vector field and together with this, with velocity, we have got energy equation which is a transport equation for a scalar field. We have got a scalar variable pressure involved in Navier-Stokes equations. For compressible flows continuity equation brings another scalar variable that is density. So here we have got a mix of the scalar and vector variables, okay.

Now, this mix allows us more freedom in the choice of grid used in numerical simulation. So what popular grid choices which have been tried, let us have a look at them. Now the choice of the grid would also depend on the discretization scheme that is to say whether we have chosen finite difference needed, finite volume method, or finite element method. We can classify these choices broadly in 2 categories based on the arrangement of our flow variables on grid nodes.

The first one we call collocated arrangement and the second one is called staggered arrangement. So what is a collocated grid. There are 2 alternative spellings which we use for collocated, either coll or c-located, the second one gives us a clearer picture, just this everything is located together. Co means together. So if all the variables that should say our velocity components, density, pressure, temperature, they are stored at same set of grid points, then the grid is called collocated grid.

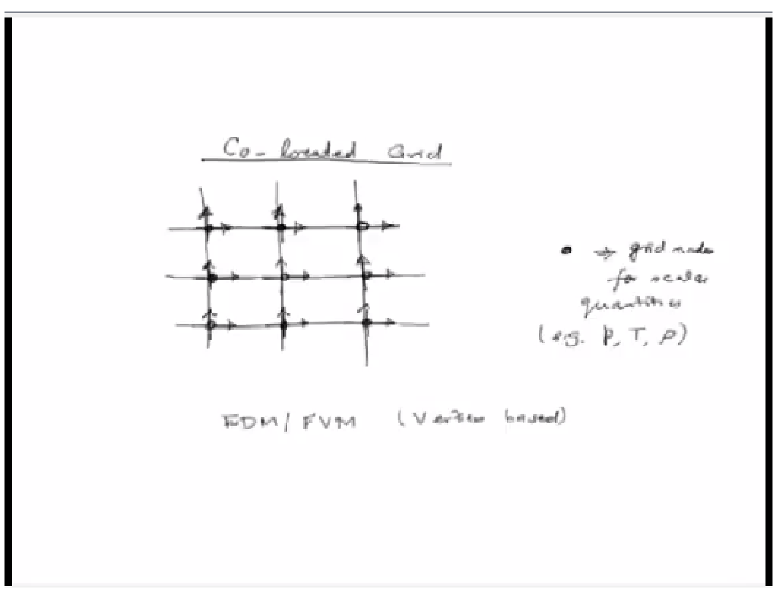
(Refer Slide Time: 21:34)



And in this case, there is considerable simplification in the programming and specifically we will not use advanced or what we call fast solvers based on the multigrid methods there. We need to use restriction and prolongation operators for hierarchy of course grids which we use. So if the same set of grid nodes are being used for all the variables, we can use the same restriction and prolongation operator for each variable, whether it is the velocity components or temperature or density.

So that are the advantage which we get in the case of co-located grids. Before proceeding further, let us have a graphical look at what do we mean by a co-located grid.

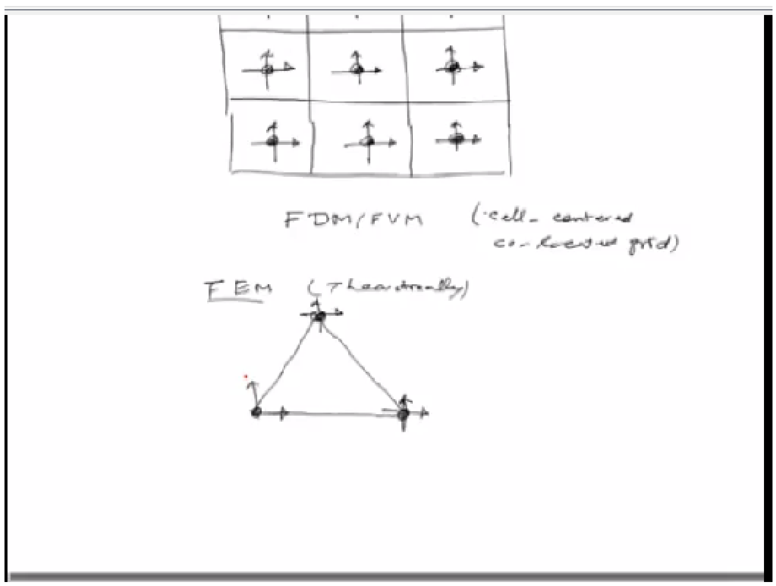
(Refer Slide Time: 22:23)



So first let us take a case of a structured grid which we might use in finite difference or finite volume formulations and suppose we take for instance, let us take vertex based grid, this is finite difference or finite volume grid vertex based. So let us use 2 symbols, we will use this field circles, field circles would represent the grid nodes for scalar quantities. For example, our pressure, temperature and density.

We are talking about co-located grid. Let the same grid points would also be used as the grid nodes for velocity. So let us use arrows to indicate, so this is X velocity component, Y velocity component. So what we say that the same grid nodes are being used to store our scalar variables as well as these vector components evenly everywhere, analogous pictures we can draw for 3-dimensional case.

(Refer Slide Time: 24:55)

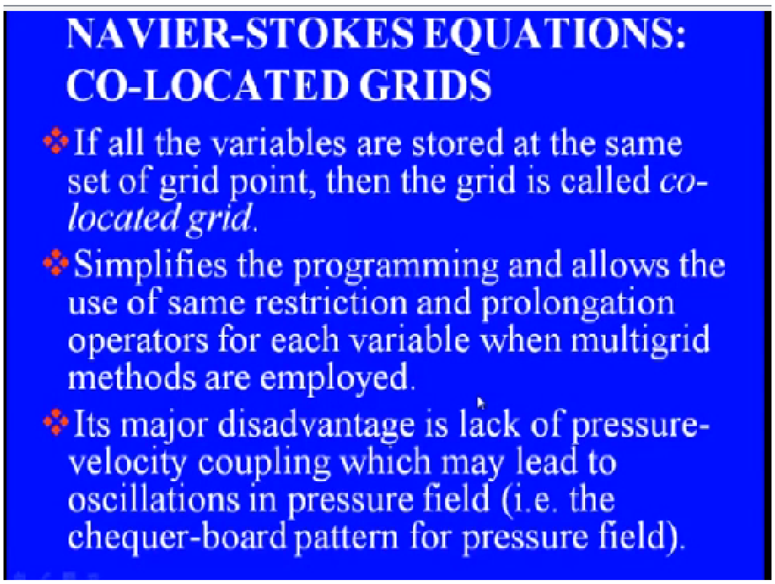


Similarly, if you are using cell-centred formulation that is our computational node is at the centre of a cell, so that is where all the other problem variables or flow variables are defined that is to say at the centroid, so field circles to represent scalar or similarly here. For velocity components also defined at these cell centroids. The situation is a bit more complicated in the case of finite elements, though theoretically this again our FDM or FVM cell centred co-located grid.

Theoretically for finite element, what we will say, let us suppose this we have got triangular elements in 2-D set, the scalars will also be defined here and our velocity components will also be defined at these nodes. Why I said theoretically because there are certain requirements which the finite element interpolations should satisfy for pressure and velocity which would not allow the use of the shape function of the same order for both pressure and velocity components.

For the sake of explanation of co-located grid in the context of finite element if you are supposed to chose, that is what we have to take there. Okay, so this co-located grid, they all have been tried, they have been popularly tried in unstructured finite volume schemes.

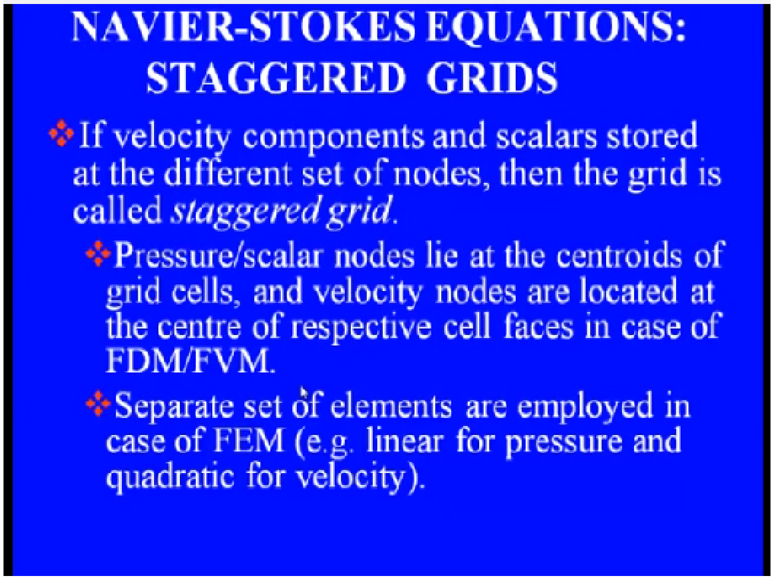
(Refer Slide Time: 27:36)



And there was a major disadvantage of these schemes, what we call lack of pressure-velocity coupling which might lead to what people call as oscillations in pressure field or chequer-board pattern for the pressure field. Now this was the situation a decade or so back when these schemes were used extensively for the first time.

The recent developments have been able to deal with this problem of chequer-board pattern and with the advent of or the popularity of what we call unstructured finite volume mesh for CFD simulations, co-located grids are back in favour in numerical simulation of Navier-Stokes equations.

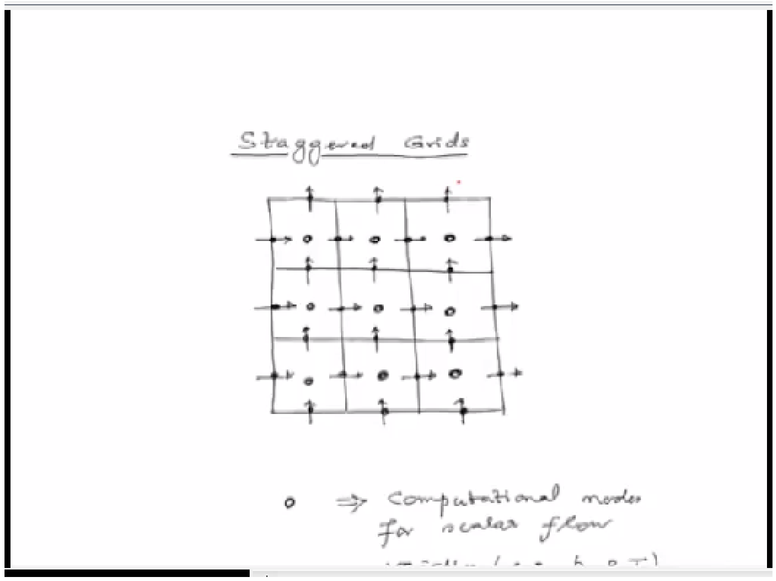
(Refer Slide Time: 28:23)



Next, let us have a look at what we call staggered grid. Now here, velocity components and the scalars are stored at different set of nodes. So if that is the situation, then we would say the grid is staggered grid. So the basic definition is that if velocity components and the scalars are stored at different set of nodes, then the grid used in CFD simulation what we called as staggered grid.

In finite difference or finite volume simulations, what we would normally choose is that the pressure or the scalar nodes, they would lie at the centroids of the grid cells and velocity nodes are located at the centre of respective cell faces in the case of finite difference or finite volume method and in the case of finite element, we will have to use separate set of elements, for instance we will use for example linear element for pressure and quadratic element for velocity.

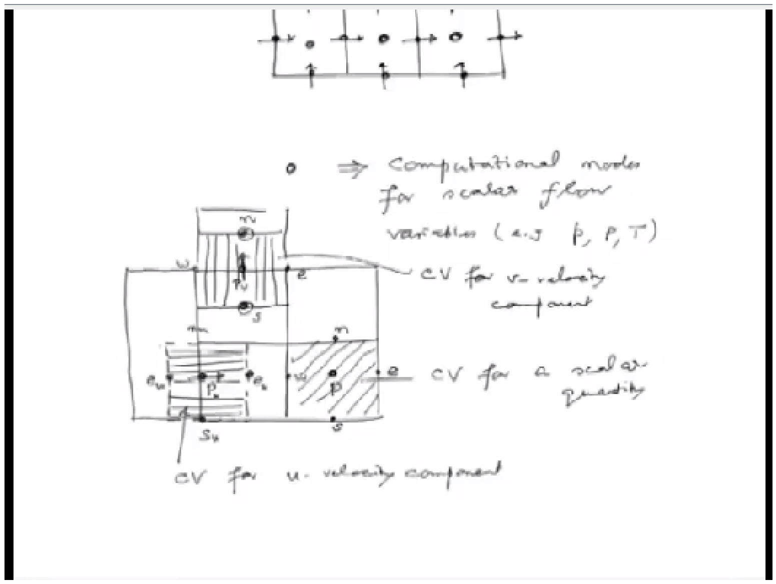
(Refer Slide Time: 29:25)



Let us have a look at the graphical representation of staggered grids. Let us draw for the sake of simplicity, a structured finite volume or finite and difference grid. Now the centroids are the ones where pressure or temperature would be defined, that is the way we are going to solve for all the scalar components. So our filled dots, they indicate computational nodes for scalar flow variables.

For example, pressure, density and temperature. The velocity components would be defined; their nodes would be defined at the cell faces. So that is where is the locations of U velocity components and similarly these faces for our V velocity components, that is where the computational node for the V velocity component would lie. Okay, if you want to have a look at how the finite volumes would look like for each of these components, let us redraw this figure and illustrate each finite volume separately.

(Refer Slide Time: 32:19)



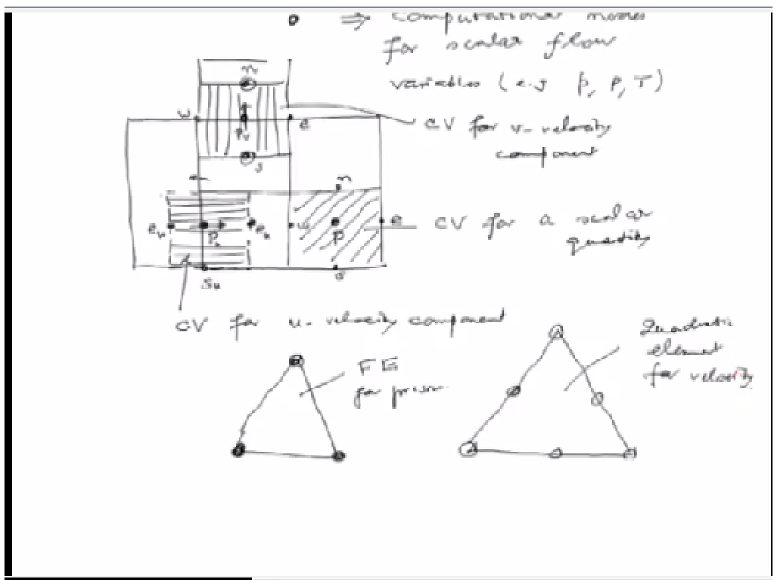
So for the scalar variables, it is very obvious, this whole cell, this is our control volume for scalar quantity. How about the U velocity component, how we will define the control volume for that, so let us draw it separately other box here. So this is where, suppose this is one of the U velocity nodes. So this corresponding control volume would be this, so with respect for instance this is P, this becomes the east face, this is our west face, south face and north face for a scalar control volume.

For control volume for U velocity component, let us call it as Pu, so this becomes our E stern face for u, western face for this. Let us hatch it differently. Similarly, southern face and this is northern face. So this is CV for U velocity component. In the same way, for the sake of clarity, let us extend this figure further here, these are scalar nodes. So the faces of V velocity component will pass through the scalar nodes.

So let us use vertical hatching. Now this is the CV for V velocity component. This is computational node on the other faces, east face, west face, south face and northern face for the control volume which we would use in finite volume formulations for V velocity component. So what we clearly say here that the computational nodes for each velocity component is at different location. In finite volume formulation, we will have to use separate finite volumes for a scalar variable and each of the velocity components.

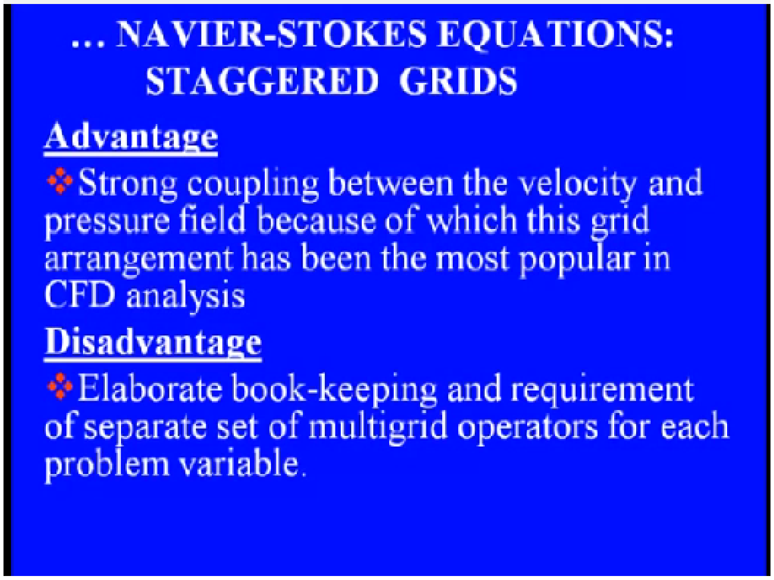
So that would introduce lots of book-keeping, it will make our algorithm part of programming component which is a bit complicated, but there is a tremendous advantage which we can see if you look at the first figure which we have here. Traditionally what we think that velocity field or the flow is driven by the difference of pressure. So velocity at any node that is because of the pressure difference at adjoining 2 computational nodes. So that is why this particular grid arrangement gives us what we call very strong pressure-velocity coupling.

(Refer Slide Time: 36:31)



Now similarly if you are dealing with finite element, then you would use linear elements for pressure. This is FE for pressure and for velocity, we will have to use the quadratic element. This is quadratic element for velocity. The advantage I just mentioned.

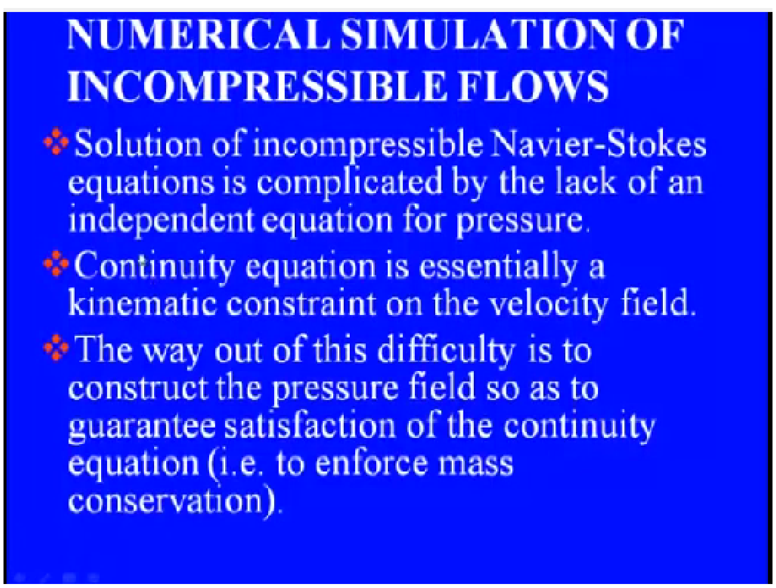
(Refer Slide Time: 37:26)



There is strong coupling between the velocity and pressure field. Because of it, this grid arrangement has been the most popular in CFD analysis and disadvantage, we have to do elaborate book-keeping and if you want to use multigrid for the solution of each of the flow variables, then we have to maintain separate set of multigrid operates but not withstanding this disadvantage of elaborate book-keeping and having separate set of operators.

the first, the advantage which is offered, that is very, very strong motivation and even today, staggered grids are used in preference to co-located grids in the context of structured finite difference or finite volume analysis.

(Refer Slide Time: 38:16)



Now let us come to the numerical simulation of incompressible flows. This one problem which we mentioned earlier, that Navier-Stokes equations for incompressible flow, they involve a term pressure gradient, the gradient of P is there, but there is no equation for pressure in contrast to compressible flows where pressure can be computed from equation of the state. We do not have any such independent equation for pressure for incompressible flow problems.

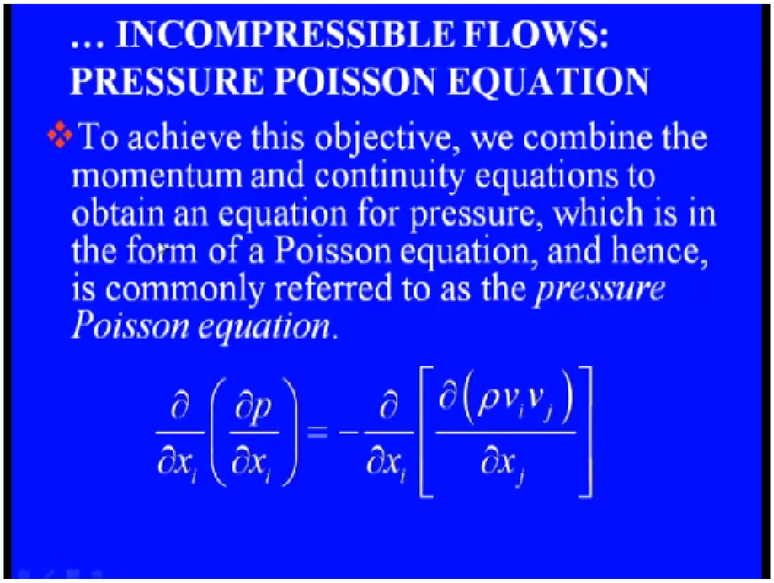
So in this case, if you do not have an equation for a variable and it is one of the most important flow variables involved which drives our flow, what do we do. So suppose we are dealing with isothermal flow, so there we will have a set of 4 equations, continuity equation, so it is divergence of V=0 and 3 equations for each of the velocity components but there is no equation for pressure.

So we have got, though unknowns are 4, pressure plus 3 velocity components and we have also got 4 equations available to us, but continuity equation cannot be used directly because it does not involve pressure, so it cannot be used as equation for pressure and it essentially represents a kinematic constraints. We have to do some manipulations to derive equation for pressure using our continuity equation.

So that is what it is where we out of this difficulty of an independent equation for pressure is to construct the pressure field so as to guarantee the satisfaction of continuity equation. So we will try to construct a pressure field whose solution would guarantee or if you use a pressure field in our momentum equation and solve for the velocity field, that velocity field is guaranteed to satisfy continuity equation.

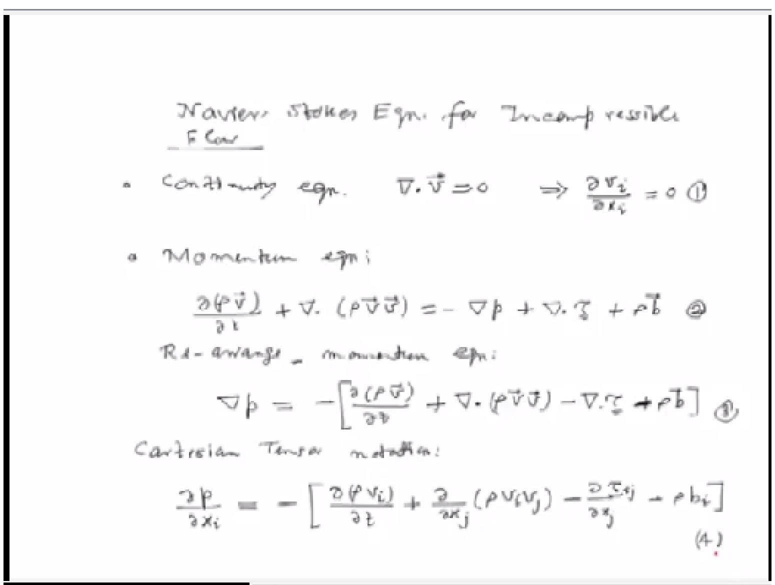
That is to say, our mass conservation is enforced. So how do we do it, how do we achieve this objective. To achieve this objective, we combine the momentum and continuity equations to obtain an equation for pressure

(Refer Slide Time: 40:42)



Which is in the form of a Poisson equation and hence it is commonly referred to as Pressure Poisson Equation. How do we obtain this equation, let us have a detailed look at it. Let us write our continuity and the momentum equations for incompressible flow.

(Refer Slide Time: 41:01)

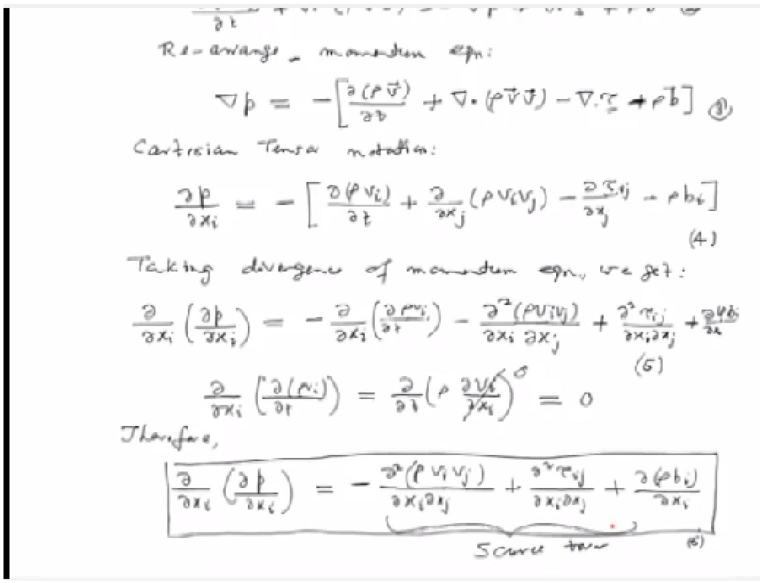


So our continuity equation is simply divergence of V=0 or in (()) (41:35) notation, we can write it is del vi/del xi=0. Next our momentum equations, del of rho v/del t+divergence of rho vv=-gradient of p+divergence of tau rho*b. Now let us rearrange the terms here. So let us rearrange, keep pressure on the left-hand side. So rearrange momentum equation, so we can write this as gradient of p=-del of rho v/del t divergence of rho vv and the remaining terms.

For simplifying our algebra, let us write this equation in Cartesian tensor notation. So left-hand side is del p/del xi=-[del of rho vi/del t, we have retained rho here, that rho is realize that we are dealing with incompressible flows, it is just a constant value, del/del xj of rho vivj del/del xj of tau ij-rho bi].

Now let us take divergence of this equation because in continuity equation, that is what we have got. So divergence operator is sitting there.

(Refer Slide Time: 44:28)

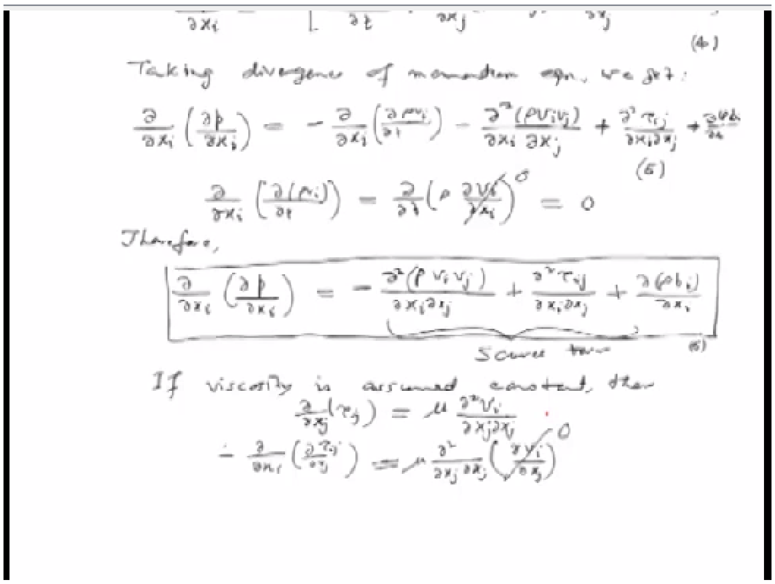


So taking divergence of momentum equation, so we will apply to this rearranged form, what do we get, del/del xi del p/del xi=-del/del xi of del/del t of rho vi-del 2 of rho vivj/del xi/del xj+del 2 tau ij/del xi del xj+del/del xi of rho bi. Now let us have a look at each term in this equation 5 separately. This del/del xi del/del t of rho vi, this we can exchange this temporal and special derivatives.

So we can write it as del/del t, rho is a constant, so we can take it out, rho del vi/del xi. Del vi/del xi that is divergence of velocity field which is 0 for incompressible flow. So this term evaluates to 0, okay. So therefore what do we get del/del xi of del p/del xi=-del 2/del xi del xj of rho vivj+del 2 tau ij/del xi del xj+del of rho bi/del xi. Now this equation has a form very similar to our Poisson equation.

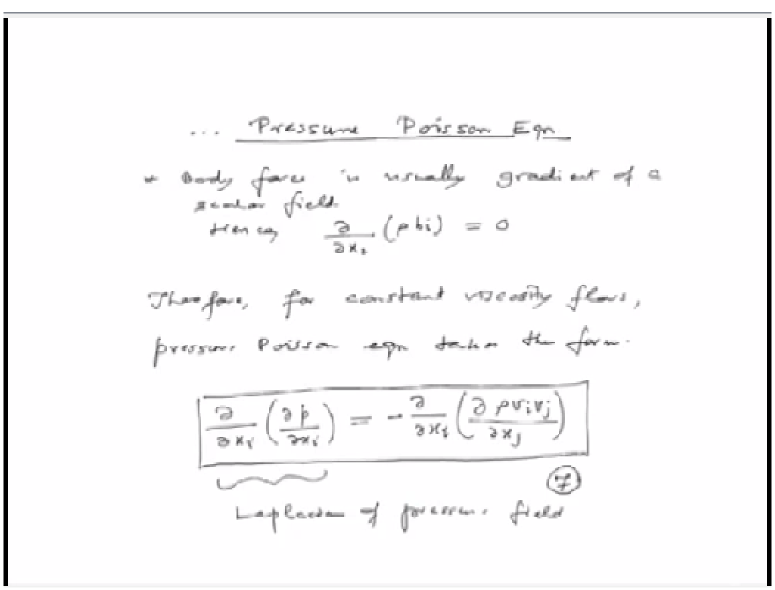
So what we have got, on the left-hand side, we have got the Laplacian for p. On the right-hand side, this represents a sort of a source term. So this is our Poisson equation, wherein we have to still maintain a possibility that viscosity may not be constant. In fact, we will get a simplified form.

(Refer Slide Time: 48:12)



So if viscosity were constant, is assumed constant, then what happens. Our del/del xj of tau ij, this reduces to mu*del 2 vi/del xj del xj. So if you apply del xi, therefore del/del xi of del/del xj tau ij, this is mu*del 2 del xj del xj of del vi/del xj. So this also vanishes because of continuity. So we get a very simple form for a Pressure Poisson equation.

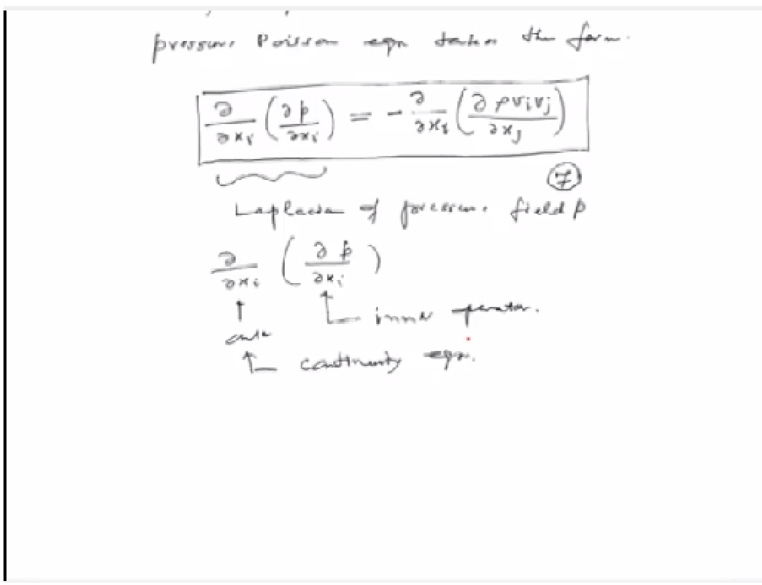
(Refer Slide Time: 49:36)



Similarly our body force term that would normally be the gradient of a scalar field, this body force is usually gradient of a scalar field, can quickly verify it for the gravitational force field which is because of the gravitational potential. Hence this divergence would be 0, so del/del xi of rho bi, this is 0. So therefore, for constant viscosity flows, that is to say if we are dealing with homogeneous isotropic and isothermal flows, our Pressure Poisson equation takes a very simple form, del/del xi del p/del xi=-del/del xi of del rho vivj and xj.

So this is our final form for the Pressure Poisson equation for flows with constant viscosity. Remember in our derivation what we have done, we have started off with momentum equation. So this inner derivative of p in our Laplacian operator, so this side, this is Laplacian of pressure field p.

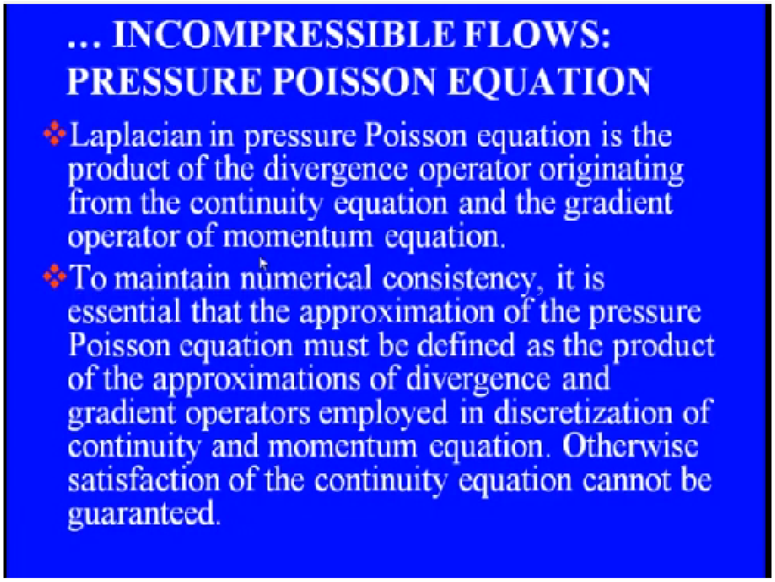
(Refer Slide Time: 52:11)



The first operator, this Laplacian we can write as del/del xi that is outer one and then we have got this inner one, del/del xi p, this inner operator. This inner operator is from a momentum equation, okay and the outer of the divergence, this comes from continuity equation because that is what motivated us to take divergence. We would apply the same operator which we would use for our continuity equation and this has got some important consequences which we must take care of in our numerical implementation.

So simplified Pressure Poisson equation, this is summary of what we have derived, del/del xi of del p/del xi= -del/del xi[del of rho vivj/del xj] and I purposely put these 2 operators separately. So that the outer del/del xi clearly indicates that this is the divergence operator coming from a continuity equation and the inner derivative del p/del xi or here del of del xj of rho vivj, these are linked to our momentum equation.

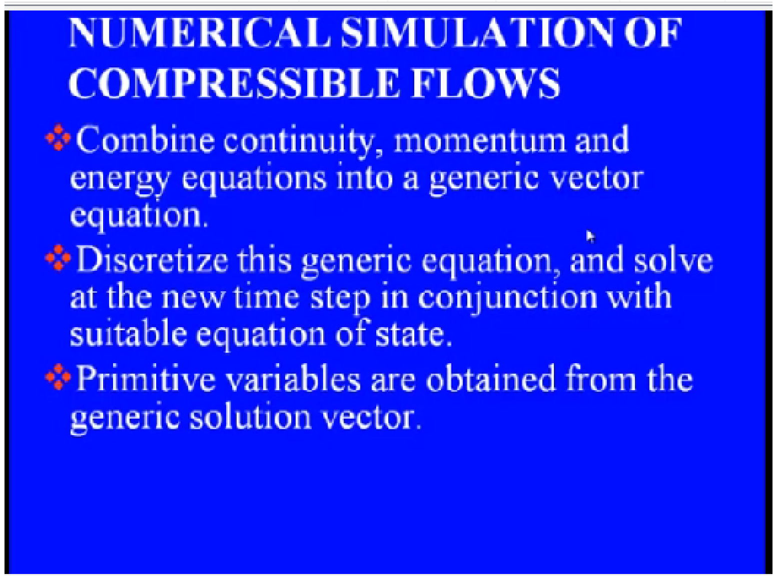
(Refer Slide Time: 53:47)



So we should be aware of it that Laplacian Pressure Poisson equation is the product of the divergence operator originating from the continuity equation and the gradient operator of the momentum equation. So to maintain numerical consistency it is essential that whatever discretization scheme we use for approximation of Pressure Poisson equation, that must be defined as a product of the approximations in divergence for continuity equations and gradient operator which we would use in our momentum equation.

If this is not taken care of, the satisfaction of continuity cannot be guaranteed by your numerical solution. We will just take a brief algorithmic look at the solution of compressible flows.

(Refer Slide Time: 54:30)

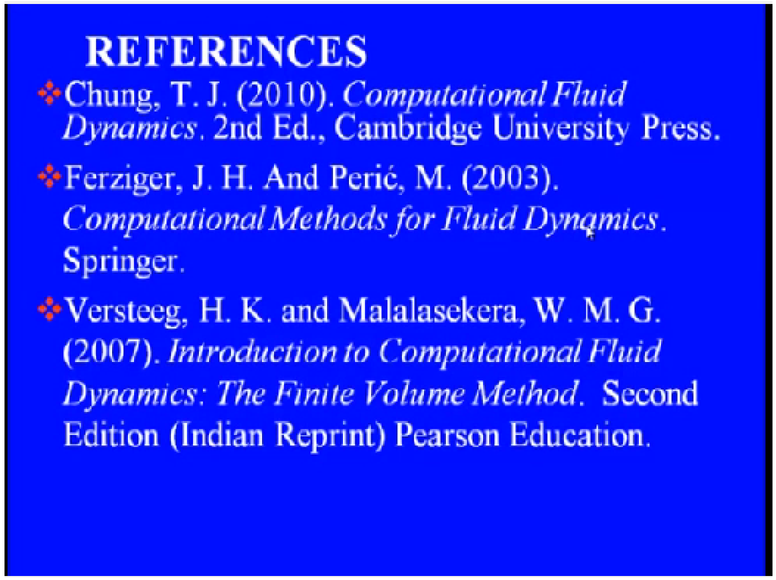


We will combine continuity, momentum and energy equations into a generic vector equations which is what is normally done and then we would discretize this generic equation usually, the simplest way would be to choose an appropriate explicit time integration scheme and obtain the solution at the next time step in conjunction with suitable equation of the state to get our pressure.

Similarly, we can also use our solution variables at a given time instant to calculate the viscosity and conductivity which is required and the primitive variables that is density, density would be solved, that would be part of one of the components of a generic vector unknown. But other components of that vector unknown would be rho U rho V and rho W and then total energy.

So the primitive variables that would be obtained from this generic solution vector rho obtained directly use or obtained by taking the ratio of rho U/rho. Similarly rho V/rho would give us V component and rho W/rho would give us the W component. Substitute for that magnitude of velocity in our total energy equation, we can get e that is internal energy, Ug=CVT, that is T=e/CV to obtain the temperature field and so on.

(Refer Slide Time: 56:14)



So we have run out of time in this lecture, may be some of the details might have a look at in future lectures time permitting. For further details for the time being, I would refer you to the references, read the book by Chung on Computational Fluid Dynamics or Computational Methods of Fluid Dynamics by Ferziger.

There is another book which is a beautiful Introduction to Computational Fluid Dynamics that is Introduction to Computational Fluid Dynamics by J. D. Anderson and that is specifically useful for compressible flow simulations. So please have a look at the solution algorithms for the compressible flows in Anderson's book.