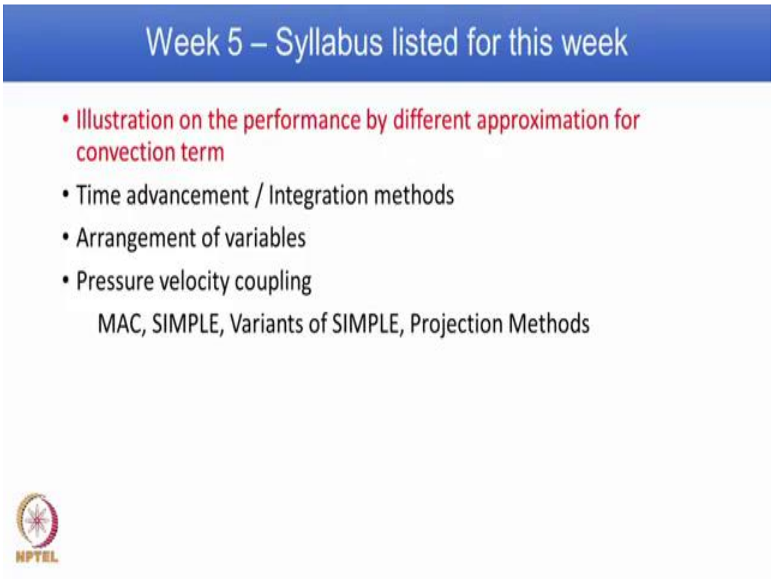
Foundation of Computational Fluid Dynamics
Dr. S. Vengadesan
Department of Applied Mechanics
Indian Institute of Technology, Madras

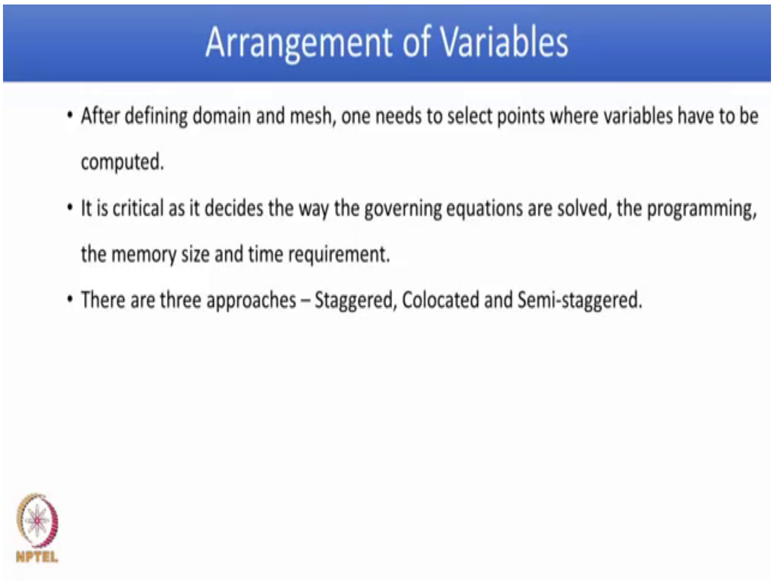
Lecture – 24

(Refer Slide Time: 00:24)



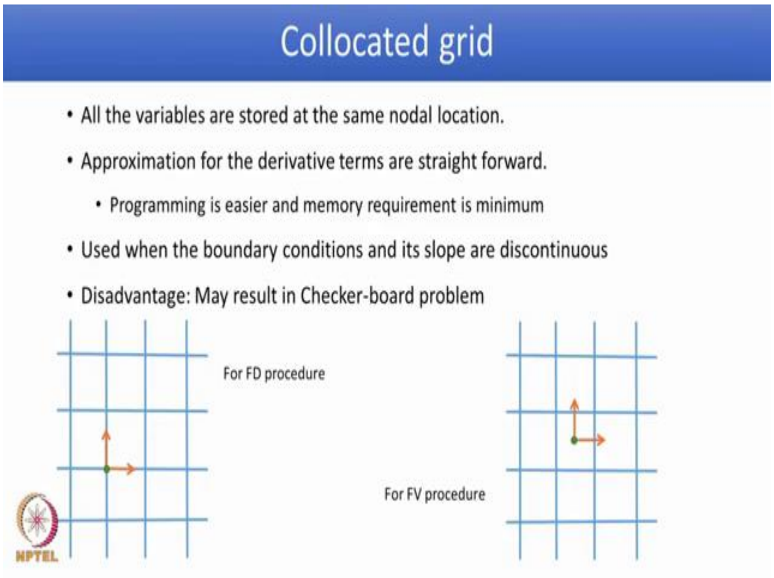
Greetings, welcome again to this course on CFD. Today we will we doing fourth module for this week. Syllabus we have listed for this week; illustration on the performance by different interpolation scheme for convection term. And from this class, we will go move onto time advancement, integration methods, arrangement of variables, pressure velocity coupling. There are different methods available MAC, SIMPLE, different versions of SIMPLE, projection methods. We will look into these details and then move onto next module.

(Refer Slide Time: 01:12)



With regards to variables, once you define domain and mesh, then we need to select points where variables have to be computed. It is critical because it decides the way the governing equations are solved, the programming easiness, memory requirement and overall computational time requirement. There are three approaches in general, what is known as a staggered grid, colocated grid, and semi-staggered grid.

(Refer Slide Time: 01:52)

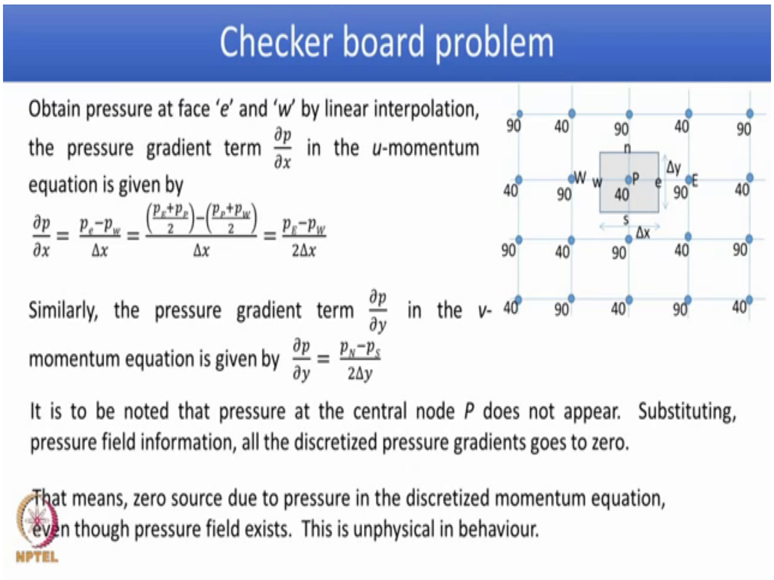


Let us look at one-by-one. The first one collocated grid, it is you can also read it as co located. As the name indicate in this system all the variables are stored at the same location. Approximation for the derivatives are straight forward; programming is easier, memory requirement is minimum. If you look at the mesh arrangement, what is shown here is for the finite difference procedure, and what is shown on the right side is for the finite volume procedure. As you can see here, this is the location, where all the variables are stored. So, in this example, the arrow going from left to right is for the u-velocity, arrow going from bottom to top is for the v-velocity, and dotted point that is marked is actually indicating node or location, where other variables are stored, scalar variables like temperature, pressure or concentration. So, all the three variables for example, u-velocity, v-velocity, pressure are stored at the same location, this is for finite difference arrangement. And you are going for finite volume type of arrangement then you can definition at the centre of the volume as shown here. So, this is the location where scalar variables are stored, and arrow going from left to right for u-velocity, arrow going from bottom to top is for v-velocity.

As you can observe in both the cases, they are defines at the same location. Let say for example, you are trying to find out first derivative dou u by dou x or the pressure dou p by dou x; both dou u by dou x and dou p by dou x appear in the u-momentum equation and for both you need to find the derivative. And in this case, as you can see here, you are defining only grid, and the grid is same for both u as well as pressure. Hence getting a derivative difference formula is easier, same coding can be used just by change the variables from velocity to pressure, you can obtain first derivative say dou u by dou x and dou p by dou x with much easier, because you are using only same location, delta x is also calculated only once and it can be stored. Hence it is easier and memory requirement is minimum.

It is used when boundary conditions as well as its slopes are discontinuous. So, this co located arrangement of variable extremely helpful when the boundary condition as well as its slope is discontinuous. Like any other method, this also has some disadvantage, what is known as checker-board problem. In the next slide, will explain what is checker-board problem.

(Refer Slide Time: 05:16)

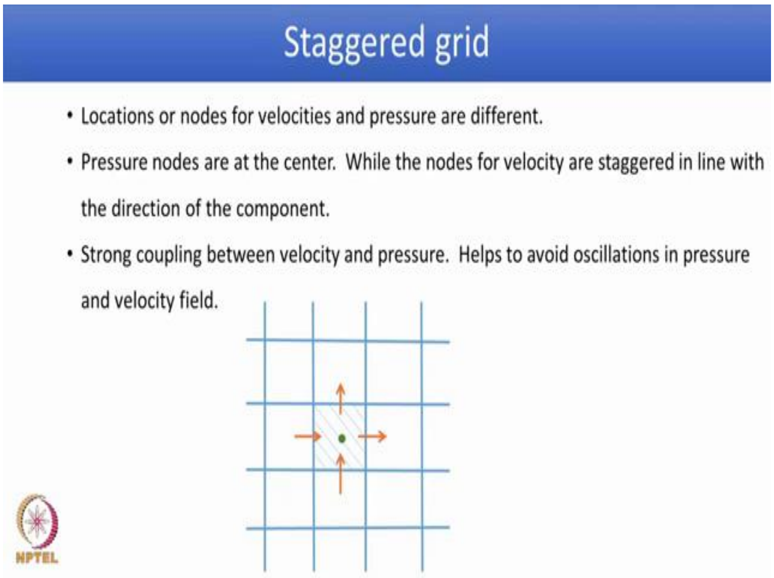


In this slide, what is shown in this figure is a computed pressure distribution on co located arrangement. A particular control volume or the pressure cell is shown in the shaded portion; P is the node of interest. For example, west node and west face, similarly east face and east nodes are shown. Similarly, for other side, that is north node, north face, south node and south face, they are also shown. Delta x and delta y is a element distance in x and y direction respectively. The numbers that is shown in this figure that is 90, 40, 90, 40, this is the pressure field obtained say by ((Refer Time: 06:06)) procedure, let us not worry about what is the procedure. If the pressure field happens to be as shown here, then how we will find out how the derivative of the pressure in x and y behave.

So, obtain pressure at face e and w by linear interpolation, the pressure gradient term dou p by dou x for example, in the u-momentum equation is shown as here. So, dou p by dou x equal to dou p by dou x for the cell that is shown it is actually P at east face minus P at w-west face by delta x that is what it is shown here, because it is dou p by dou x. Now, P at east face is done by linear interpolation, pressure at node P – point of interest, and pressure at node E – the adjacent node are used to get pressure at east face. So, it is shown here P at capital E plus P at P by 2. Similarly, for P at west face, you take pressure information from node interest P, and west node and linear interpolation is used delta x is there. Now, if you look at this overall, you have P p appearing in both the term with the minus in between, so P p term gets cancelled. Finally, you have p at e minus p at w by two delta x.

You can obtain similarly the other pressure gradient term dou p by dou y, which is appearing in second momentum equation or v-momentum equation. Without going to the details, we can immediately make a guess dou p by dou y is equal to p at N minus p at S by two delta y. Now, what is to be noted you are interested to find out dou p by dou x and dou p by dou y and you get expression without the node of interest term appearing. So, it is to be noted that pressure at the central node P does not appear in the final expression for this gradient dou p by dou x or dou p by dou y. Now, if you substitute the pressure gradient term in discretized Navier-Stokes equation, then all the pressure gradient term will go to zero, which otherwise means there is no source term appearing in the discretized Navier-Stokes equation, which is unphysical. So, that means, the zero source due to pressure in the discretized momentum equation, even though there is a pressure field appearing as shown in this figure. And this is quite unphysical and this is the problem with collocated arrangement.

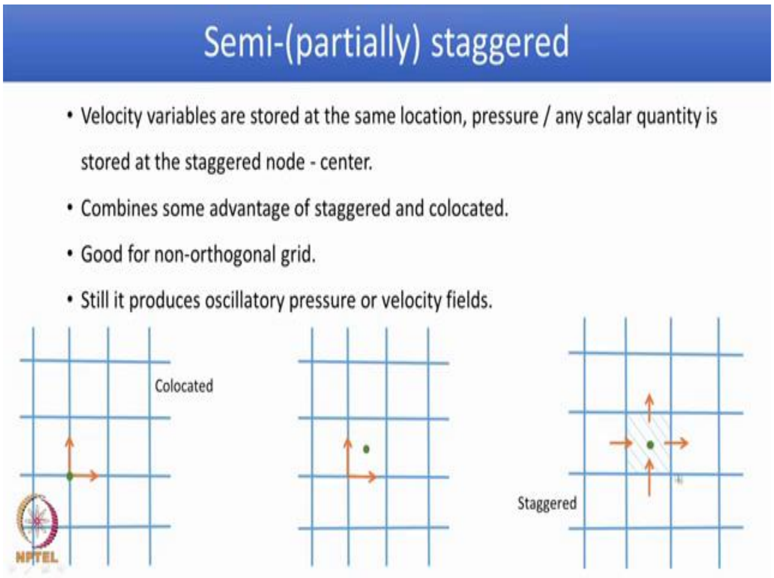
(Refer Slide Time: 09:01)



Next arrangement is the staggered arrangement as a name indicate variables are stored at staggering location, location or nodes where velocities and pressure are different, not only pressure any scalar also different. So, here is the figure to explain what is the staggered arrangement, for the same mesh, you can now observe at the centre of the mesh is the pressure or any scalar variable is stored; arrow mark going from left to right is indication for u velocity; and arrow mark going from bottom to top is for the v velocity. So, u velocity variables are stored on the vertical line at this place, which is half cell distance from the center in the x-direction. Similarly, v-velocity cells are stored on the horizontal line as shown here, which is half- cell distance in the y-direction from the center of the cell as shown here.

So, you have three locations, one for each velocity, and one for pressure or scalar; whereas, in the co-located arrangement, we have the same location. Now, this results in different computation. Pressure nodes are at the center. While the nodes for velocity are staggered in line with the direction of the component. There is one big advantage, there is a strong coupling between velocity and pressure, so as you can interpret from this figure, the pressure at the center is responsible for the u-velocity flow as well as for the v-velocity flow as shown in the figure. Hence there is a natural strong coupling between velocity and pressure, and this avoids oscillations in pressure as well as velocity field.

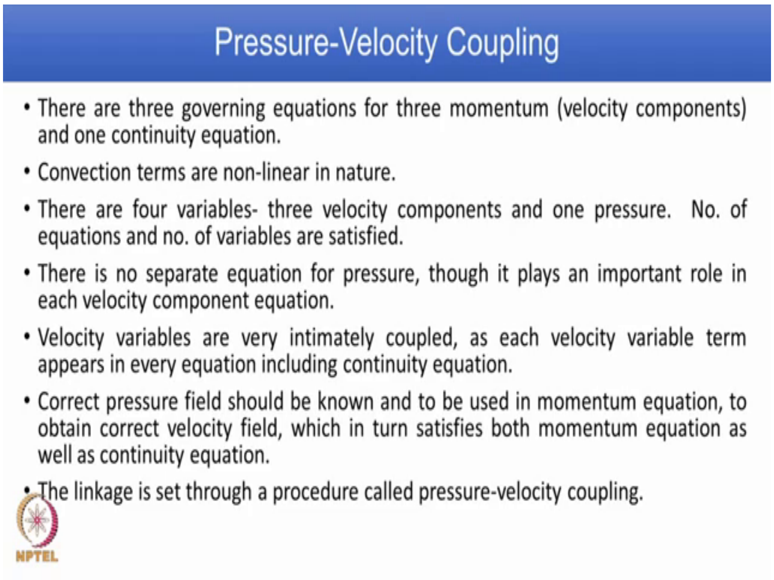
(Refer Slide Time: 11:00)



Third arrangement is what is known as semi or partially staggered arrangement. In this, velocity variables are stored at the same location; whereas, pressure or any other scalar quantity is stored at the staggered node or at the center. So, this is the figure that is explained in for the same mesh arrangement, u-velocity and v-velocity are stored at one same location, whereas, pressure is at the center. So, it is in between colocated and staggered. It combines advantage of staggered as well as colocated and it is found to be good for non-orthogonal grid. Still it produces some oscillatory pressure or velocity fields.

Now, let us look at all the three arrangements together, the left is the colocated arrangement, middle is the semi-staggered or partial arrangement, in the right is the staggered arrangement, so you can immediately get idea the advantage as well as disadvantage. So, in the case of staggered arrangement, because locations are different, you need to compute for each one of them separately hence it involved sometimes and more memory requirement. Whereas, there is natural pressure and velocity coupling in staggered grid arrangement. In the next example, we are going to use only staggered grid arrangement and that is widely used in many codes as well commercial software.

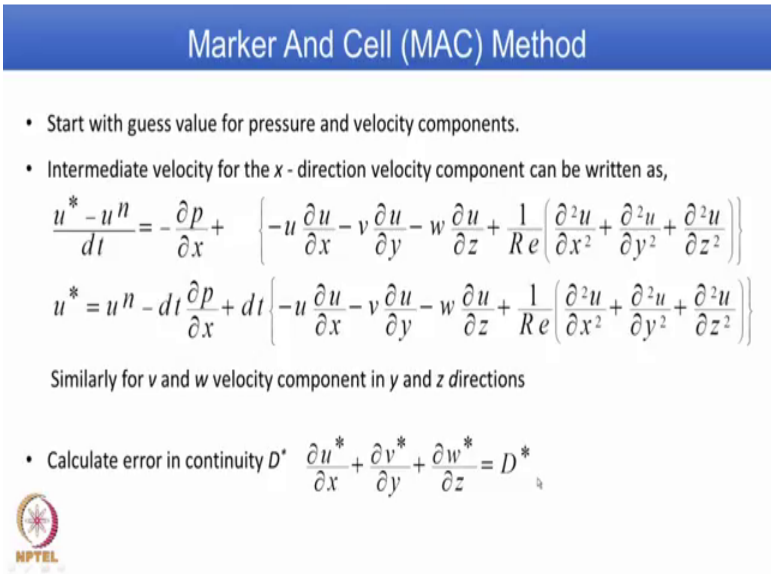
(Refer Slide Time: 12:35)



We go to the next important topic what is known as pressure-velocity coupling. What is the background behind pressure-velocity coupling or what do we mean by pressure-velocity coupling. By now we know in general there are three governing equations respectively for three momentum or three velocity components, and there is one continuity equation, this is the minimum requirement for any fluid flow calculations. In each governing equation for momentum, there is a convection term and we repeat it many times before convection terms are non-linear in nature. In total, there are four variables that is there are three velocity components and one pressure. The number of equations equal to number of variables is satisfied.

However, there is no separate equation for pressure, though it plays an important role in all the momentum equation. Velocity variables are; however, very strongly coupled, because you will observe you know already also that each velocity component appears in every other momentum equation, including the continuity equation that way velocity variables are very intimately coupled. Correct pressure field should be known and that should be used in momentum equation, to obtain correct velocity field, which in turn satisfies both momentum as well as continuity equation. This process of linkage within momentum equation and continuity equation is what is known as a pressure-velocity coupling. Now, this is very important stage in solving Navier-Stokes equation, there are many methods available, we are going to look at specifically three, four method MAC algorithm, SIMPLE algorithm, SIMPLER algorithm, projection methods.

(Refer Slide Time: 14:44)

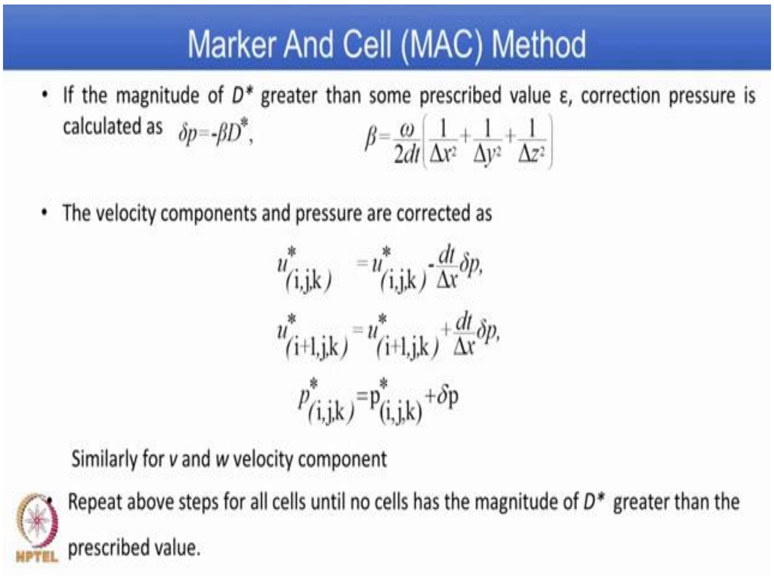


Let us look at MAC algorithm first; this is otherwise called Marker and Cell method. For any calculation, once you define mesh and you decide the variable location, you are able to describe boundary conditions, but what about all the internal nodes. All the internal nodes are initialized with some guess values, this step is called initialization. So, you start the solution with the guess value of pressure and velocity component. We will now explain the procedure for u-momentum equation, the procedure is same for v as well as w momentum equation; between time level zero or between one iteration to the next iteration, the velocity is intermediate velocity. Once you know the procedure for discretization then you write down final discretized Navier-Stokes equation. In this case, we mentioned, we will take x-momentum equation and that is what is shown here.

So, the time derivative term is retained on the left side, all other terms are brought to the right side, which means all the terms are supposed to be known right side. U star is a intermediate velocity and that is to be determined first then it will be corrected. So, u star minus u n by dt that is the time integration; for this case, what is shown here is a first order Euler time integration; u with the superscript n refers to previous time level. All the terms on the right side dou p by dou x and convection term, viscous diffusion term, we follow any procedure for discretization, finally, we get this discretized equation. So, if you solve this equation, then you get u star, we name it as u star, because it is an intermediate velocity. The intermediate velocity may not satisfied continuity equation.

So, if you rewrite this equation, take d p to the other side; u n is also known and that is also taken to the other side, and this is the equation for u star. So, if you solve, you get u star, and this is at one node location, and for one-d, it will run from left to right; for two-d, it will run the entire map. You can write the similar expression for v as well as w velocity component for y as well as z direction. As we mentioned just now, this u star, v star, w star are intermediate velocity component, and they will not satisfy or they may not satisfy continuity equation. So, if you substitute u star, v star, w star in continuity then you get what is known as a error in the continuity and it is referred here as D star, you write in the form of equation as shown here. So, dou u star by dou x plus dou y star by dou y plus dou w star by dou z equal to D star. D star is a error in the continuity.

(Refer Slide Time: 18:04)



And this D star is used to obtain what is known as a correction in the pressure. And you prescribe a limit for D star; if D star is greater than some prescribed value epsilon, for example, 10 power minus 3 or 10 power minus 5 that is the tolerance level you can accept. This limit is set by you, and it will change from problem to problem. You can also be change in one calculations itself between different time level of the calculation. So, if the magnitude D star is greater than some prescribed value epsilon then you setup pressure correction as shown, delta p is correction in pressure equal to minus beta D star. And what is beta, beta is defined as shown here, omega by 2 delta t 1 by del x square plus 1 by delta y square plus 1 by delta z square. Now, omega in this case is what is known as relaxation parameter, we will talk about relaxation parameter later. So, once you calculate beta, because all the quantities are known then you can calculate delta p, delta p is the change in pressure or correction in pressure.

Velocity components and pressure are corrected as given here. What is shown here is again only for u velocity component, so u star i comma j comma k, we are writing it for three dimension, hence we get three subscripts equal to u star i comma j comma k minus dt by delta x and delta p. The adjacent node also should be corrected that is u star you can see i plus one comma j comma k equal to the old one plus the correction in pressure. Similarly, the pressure is also corrected. So, in this equation, u star that is on the left side is the new velocity, p star is the new pressure, you can use them as a u star or you can write it as u itself, it is immaterial. What is to be noted is this is the corrected value and corrected pressure.

One can repeat this procedure for other two component v as well as w. Now, repeat this step now for all the cells in computational domain until no cells has the magnitude greater than D star that you have specified. So, this procedure is what is known as Marker and Cell method, and this is again a popular method. So, this class we have specifically seen arrangement of variables, three arrangement of variables, advantage and disadvantage associated with them. Then we understood the need for pressure-velocity coupling. We explained one procedure what is known as a Marker and Cell method. In the next class, we will take another important method what is known as a SIMPLE method.

Thank you and see you all again next class.