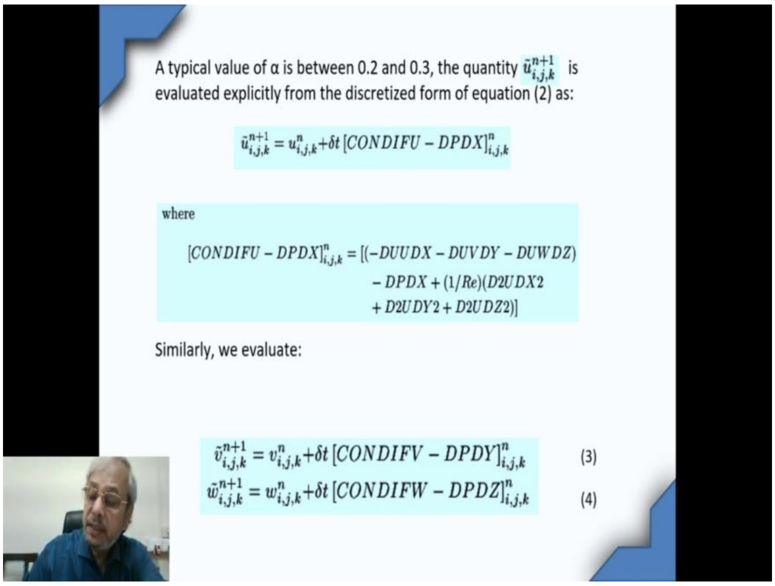
Computational Fluid Dynamics and Heat Transfer
Prof. Gautam Biswas
Department of Mechanical Engineering
Indian Institute of Technology, Kanpur

Lecture - 20
Solution of N - S equations for Incompressible Flows Using MAC Algorithm

Good morning, everybody, today, we will continue with MAC Algorithm and we will start our lecture from where we finished the last lecture.

(Refer Slide Time: 00:26)



A typical value of α is between 0.2 and 0.3, the quantity $\tilde{u}^{n+1}_{i,j,k}$ is evaluated explicitly from the discretized form of equation (2) as:

$$\tilde{u}^{n+1}_{i,j,k} = u^{n}_{i,j,k} + \delta t\,[CONDIFU - DPDX]^{n}_{i,j,k}$$

where

$$[CONDIFU - DPDX]^{n}_{i,j,k} = [(-DUUDX - DUVDY - DUWDZ) - DPDX + (1/Re)(D2UDX2 + D2UDY2 + D2UDZ2)]$$

Similarly, we evaluate:

$$\tilde{v}^{n+1}_{i,j,k} = v^{n}_{i,j,k} + \delta t\,[CONDIFV - DPDY]^{n}_{i,j,k} \qquad (3)$$

$$\tilde{w}^{n+1}_{i,j,k} = w^{n}_{i,j,k} + \delta t\,[CONDIFW - DPDZ]^{n}_{i,j,k} \qquad (4)$$

If you recall, in the last lecture, we discussed about advancement of the velocity components in each cell from n^{th} level to a level $n+1$ and these advancements are done in an explicit manner. As we can see here, $\tilde{u}^{n+1}_{i,j,k} = u^{n}_{i,j,k} + \delta t[CONDIFU - DPDX]^{n}{}_{i,j,k}]$. So, $CONDIFU$ minus $DPDX$ this means convective and diffusive terms and the pressure gradient term.

So, these ($CONDIFU$ and $DPDX$) are:

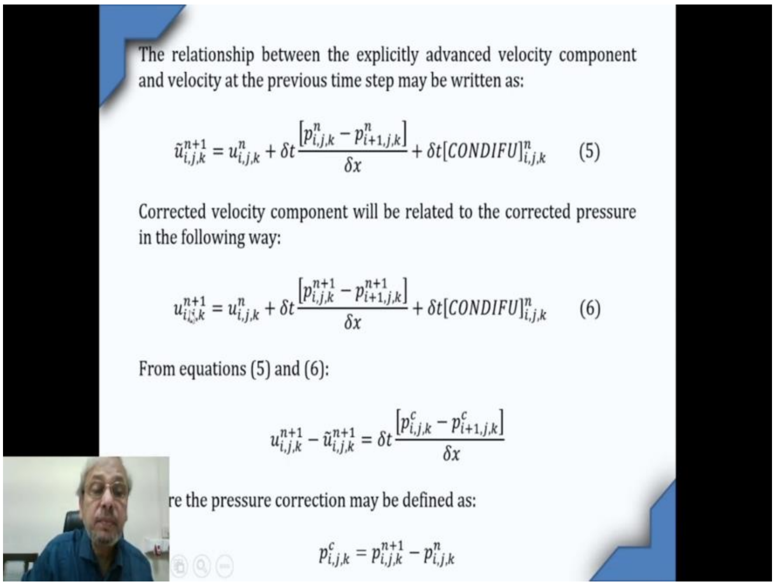
$$[CONDIFU - DPDX]^{n}_{i,j,k} = [(-DUUDX - DUVDY - DUWDZ) - DPDX + (1/Re)(D2UDX2 + D2UDY2 + D2UDZ2)] \qquad (2)$$

and these terms (right hand side of eq. 2) are being added with after having multiplied with δt, being added with $u_{i,j,k}^{n}$ to yield $\tilde{u}_{i,j,k}^{n+1}$ but, this is the I mean this ($\tilde{u}_{i,j,k}^{n+1}$) level components, velocity components have not yet satisfied continuity equation in each cell.

So, we are calling these velocities as provisionally advanced velocities. So, instead of writing $u_{i,j,k}^{n+1}$, we are also using a **tilde** sign that means, it is not yet $u_{i,j,k}$ at $n+1$, it is provisionally advanced $\tilde{u}_{i,j,k}^{n+1}$. So, in the similar way, we can calculate $\tilde{v}_{i,j,k}^{n+1}$, but provisional.

So, provisionally advanced v velocity ($\tilde{v}_{i,j,k}^{n+1}$), provisionally advanced w velocity ($\tilde{w}_{i,j,k}^{n+1}$) and provisionally advanced v and w velocities, we will have corresponding convective diffusive terms and the pressure gradients. So, this way, we advance the velocity components or the velocities at each cell from a level n to $n+1$ but these velocities (provisional velocities) have not yet satisfied continuity equation in each cell. So, we are calling them provisionally advanced velocities.

(Refer Slide Time: 04:01)



The relationship between the explicitly advanced velocity component and velocity at the previous time step may be written as:

$$\tilde{u}_{i,j,k}^{n+1} = u_{i,j,k}^{n} + \delta t \frac{\left[p_{i,j,k}^{n} - p_{i+1,j,k}^{n}\right]}{\delta x} + \delta t [CONDIFU]_{i,j,k}^{n} \qquad (5)$$

Corrected velocity component will be related to the corrected pressure in the following way:

$$u_{i,j,k}^{n+1} = u_{i,j,k}^{n} + \delta t \frac{\left[p_{i,j,k}^{n+1} - p_{i+1,j,k}^{n+1}\right]}{\delta x} + \delta t [CONDIFU]_{i,j,k}^{n} \qquad (6)$$

From equations (5) and (6):

$$u_{i,j,k}^{n+1} - \tilde{u}_{i,j,k}^{n+1} = \delta t \frac{\left[p_{i,j,k}^{c} - p_{i+1,j,k}^{c}\right]}{\delta x}$$

re the pressure correction may be defined as:

$$p_{i,j,k}^{c} = p_{i,j,k}^{n+1} - p_{i,j,k}^{n}$$

So, if we really analyze the situation, what we have done? We have basically multiplied contributions of convective and diffusive components of U-momentum equation with δt and then, we have added the pressure gradient term so, minus $DPDX$ which is $\frac{p_{i,j,k}^{n} - p_{i+1,j,k}^{n}}{\delta x}$ minus sign.

So, $-DPDX$ we are writing in this way $\left(\frac{p_{i,j,k}^{n}-p_{i+1,j,k}^{n}}{\delta x}\right)$, we are bringing in plus sign here (in Eq. 5) and so, and that $DPDX$ is also being multiplied by δt so, this is multiplied by δt to produce basically, the $\tilde{u}_{i,j,k}^{n+1}$, but we are using tilde sign because the pressures that we have used in calculating u velocity, those pressures are at a level n, they are not at a level $n+1$. So, basically there are two discrepancies, one is these velocity quantities $(\tilde{u}_{i,j,k}^{n+1}, \tilde{v}_{i,j,k}^{n+1}\ and\ \widetilde{w}_{i,j,k}^{n+1})$, we are calling them provisional why? They have not satisfied continuity equation. Number two that, both the observations are complementary to each other. Number two is that while calculating this provisionally advanced velocity quantities, we have used pressure at a level n, we have no other options, we have to use that, but since we are using pressure at a level n, we have to call these something other than $n+1$ which is $n+1$ tilde.

Because in the ideal situation, these pressures (refer Eq. 5 in slide time 4:01) should be at a level say at a same level of the velocity. So, that is what is observation in equation 5 and again, I am just restating that $-DPDX$ we are writing in this way $\left(\frac{p_{i,j,k}^{n}-p_{i+1,j,k}^{n}}{\delta x}\right)$, we are bringing in plus sign here so, within $DPDX$, it is $\left(\frac{p_{i,j,k}^{n}-p_{i+1,j,k}^{n}}{\delta x}\right)$. Originally it was $\left(\frac{p_{i+1,j,k}^{n}-p_{i,j,k}^{n}}{\delta x}\right)$. Since we are changing the sign, it is now $\left(\frac{p_{i,j,k}^{n}-p_{i+1,j,k}^{n}}{\delta x}\right)$. So, this is $DPDX$ into δt.

Now, what should have been the ideal situation, let us discuss that. Ideal situation should have been these pressure quantities $(p_{i,j,k}^{n}\ and\ p_{i+1,j,k}^{n})$ instead of remaining at a level n, these pressures should have been at a level $n+1$, then we could have written $u_{i,j,k}^{n+1}$. We did not have to use tilde sign and here, we are not using tilde sign.

So, if the pressures at a level $n+1$, these velocities will evolve as velocities at a level $n+1$ not provisional velocities at $n+1$. These are $(\tilde{u}_{i,j,k}^{n+1}, \tilde{v}_{i,j,k}^{n+1}\ and\ \widetilde{w}_{i,j,k}^{n+1})$ provisional because these have not satisfied continuity and another reason is these have used pressures which are at a level n.

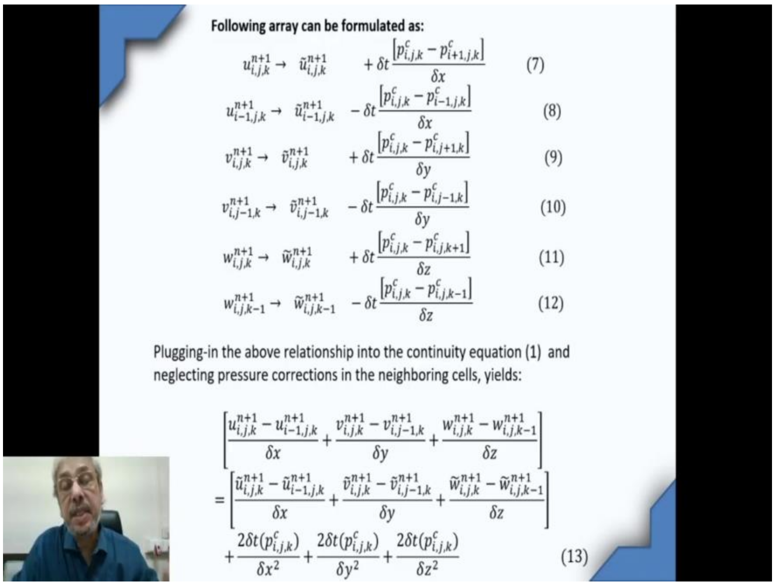
So, conceptually, this is the correct equation. Although, we are using or calculating provisional velocities equation 5. So, there is a difference between equation 5 and 6. Now, this difference if we want to really estimate, it is basically then:

$$u_{i,j,k}^{n+1} - \tilde{u}_{i,j,k}^{n+1} = \frac{\delta t\left[p_{i,j,k}^{c} - p_{i+1,j,k}^{c}\right]}{\delta x}$$

Simply $p_{i,j,k}^{c}$ “c” in pressure correction. Pressure correction is at each cell is pressure at $(n+1)^{th}$ level minus pressure at n^{th} level.

So, $p_{i,j,k}^{c} = p_{i,j,k}^{n+1} - p_{i,j,k}^{n}$. So, $p_{i,j,k}^{c}$ is the pressure correction at i,j,k cell and if these pressures are corrected, then instead of $p_{i,j,k}^{n}$, we will have $p_{i,j,k}^{n+1}$. So, we can now establish this relationship that is provisionally advanced velocity and the velocity which is really velocity at $(n+1)^{th}$ level is δt into the pressure correction by δx. Pressure correction at i,j,k cell minus pressure correction at $i+1,j,k$ cell divided by δx. So, we can establish this relationship.

(Refer Slide Time: 11:04)



Following array can be formulated as:

$$u_{i,j,k}^{n+1} \rightarrow \tilde{u}_{i,j,k}^{n+1} \quad + \delta t\frac{\left[p_{i,j,k}^{c} - p_{i+1,j,k}^{c}\right]}{\delta x} \qquad (7)$$

$$u_{i-1,j,k}^{n+1} \rightarrow \tilde{u}_{i-1,j,k}^{n+1} \quad - \delta t\frac{\left[p_{i,j,k}^{c} - p_{i-1,j,k}^{c}\right]}{\delta x} \qquad (8)$$

$$v_{i,j,k}^{n+1} \rightarrow \tilde{v}_{i,j,k}^{n+1} \quad + \delta t\frac{\left[p_{i,j,k}^{c} - p_{i,j+1,k}^{c}\right]}{\delta y} \qquad (9)$$

$$v_{i,j-1,k}^{n+1} \rightarrow \tilde{v}_{i,j-1,k}^{n+1} \quad - \delta t\frac{\left[p_{i,j,k}^{c} - p_{i,j-1,k}^{c}\right]}{\delta y} \qquad (10)$$

$$w_{i,j,k}^{n+1} \rightarrow \tilde{w}_{i,j,k}^{n+1} \quad + \delta t\frac{\left[p_{i,j,k}^{c} - p_{i,j,k+1}^{c}\right]}{\delta z} \qquad (11)$$

$$w_{i,j,k-1}^{n+1} \rightarrow \tilde{w}_{i,j,k-1}^{n+1} \quad - \delta t\frac{\left[p_{i,j,k}^{c} - p_{i,j,k-1}^{c}\right]}{\delta z} \qquad (12)$$

Plugging-in the above relationship into the continuity equation (1) and neglecting pressure corrections in the neighboring cells, yields:

$$\left[\frac{u_{i,j,k}^{n+1} - u_{i-1,j,k}^{n+1}}{\delta x} + \frac{v_{i,j,k}^{n+1} - v_{i,j-1,k}^{n+1}}{\delta y} + \frac{w_{i,j,k}^{n+1} - w_{i,j,k-1}^{n+1}}{\delta z}\right]$$
$$= \left[\frac{\tilde{u}_{i,j,k}^{n+1} - \tilde{u}_{i-1,j,k}^{n+1}}{\delta x} + \frac{\tilde{v}_{i,j,k}^{n+1} - \tilde{v}_{i,j-1,k}^{n+1}}{\delta y} + \frac{\tilde{w}_{i,j,k}^{n+1} - \tilde{w}_{i,j,k-1}^{n+1}}{\delta z}\right]$$
$$+ \frac{2\delta t(p_{i,j,k}^{c})}{\delta x^2} + \frac{2\delta t(p_{i,j,k}^{c})}{\delta y^2} + \frac{2\delta t(p_{i,j,k}^{c})}{\delta z^2} \qquad (13)$$

And we have exactly repeated that $p_{i,j,k}^{n+1} - p_{i,j,k}^{n}$ is $p_{i,j,k}^{c}$ and they are related pressure correction at $p_{i,j,k}^{c}$ minus pressure correction at $p_{i+1,j,k}^{c}$, they are related to the final velocity minus the provisionally advance velocity quantity.

So, this provisionally advance velocity quantity and final by you know quantity which we should have obtained, they are related through this $\left(u_{i,j,k}^{n+1} - \tilde{u}_{i,j,k}^{n+1} = \frac{\delta t\left[p_{i,j,k}^{c} - p_{i+1,j,k}^{c}\right]}{\delta x}\right)$ relationship with the pressure and then, we can write $u_{i,j,k}^{n+1}$ will tend to we have not because it may not be immediately you know evolve to that level because you know this pressure corrections have to be correct, then only we will get $p_{i,j,k}^{n}$ and we may require to level you know repeat the calculation. So, we are writing $u_{i,j,k}^{n+1}$, this will evolve if we keep on adding with $\tilde{u}_{i,j,k}^{n+1} + \frac{\delta t\left[p_{i,j,k}^{c} - p_{i+1,j,k}^{c}\right]}{\delta x}$ and this $p_{i,j,k}^{c}$ and $p_{i+1,j,k}^{c}$ are pressure correction terms.

Similarly, then, we can you know write for the velocity quantity at $i-1,j,k$ cell. So, $u_{i-1,j,k}^{n+1}$ is again $\tilde{u}_{i-1,j,k}^{n+1} - \frac{\delta t\left[p_{i,j,k}^{c} - p_{i-1,j,k}^{c}\right]}{\delta x}$ you can easily you know do this calculation and get this expression.

Now, we also look at $v_{i,j,k}^{n+1}$ and we will provisionally advance v velocity and that should be corrected this way (Eq. 9 in Slide Time: 11:04) through the pressure correction terms. Now, $v_{i,j-1,k}^{n+1}$, this is basically again provisionally advanced velocity at $\tilde{v}_{i,j,k}^{n+1}$ and this is the correction term.

This is a correction term we are using pressure correction effectively; this level complete quantity is velocity correction. So, these pressure connections are contributing to correction for the velocity.

So, similarly $w_{i,j,k}^{n+1}$ is provisionally advanced $\widetilde{w}_{i,j,k}^{n+1}$ velocity plus velocity correction which is related to the pressure correction term and $w_{i,j,k-1}^{n+1}$, this (Eq. 12 in Slide Time: 11:04) is provisionally advanced velocity at $\widetilde{w}_{i,j,k-1}^{n+1}$ plus the velocity correction which is $-\frac{\delta t\left[p_{i,j,k}^{c} - p_{i,j,k-1}^{c}\right]}{\delta z}$.

So, in each, if we consider a cell which has the proper you know continuity cell, if you recall that we discussed at the in the first lecture so, that cell this is staggered rate arrangement on the eastern phase, we will have this $(u_{i,j,k}^{n+1})$ velocity, on the western phase, we will have this velocity $(u_{i-1,j,k}^{n+1})$. On the northern phase, we will have this velocity $(v_{i,j,k}^{n+1})$ and on the southern phase, we will have this velocity $(v_{i,j-1,k}^{n+1})$.

And one of the sides will have this w component of velocity ($w_{i,j,k}^{n+1}$) and just the opposite phase of that side will have w component of velocity defined by this ($w_{i,j,k-1}^{n+1}$), and these are the velocities on each phase of a cell and basically, when we write continuity equation for a cell, we evolve all these east, west, north, south and two side walls in such a way that we write the difference between them to set up the gradients to write the $\frac{\partial u}{\partial x}$ term $\frac{\partial v}{\partial y}$ term and $\frac{\partial w}{\partial z}$ term. So, plugging-in the above relationship into continuity equation 1 which $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$ and neglecting the pressure corrections in the neighboring cells. So, you just substitute into $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$, these u components and v components, w components and $\delta x, \delta y\ and\ \delta z$ you will get on the left-hand side, this quantity.

And $(n+1)^{th}$ level, this must satisfy the continuity equation means this is a continuity equation and this should be equal to 0 ($\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$), but now, instead of becoming 0, the right-hand side will be all these terms (Right hand side of Eq. 13 in Slide Time: 11:04). So, basically, again, we can reorganize i.e.

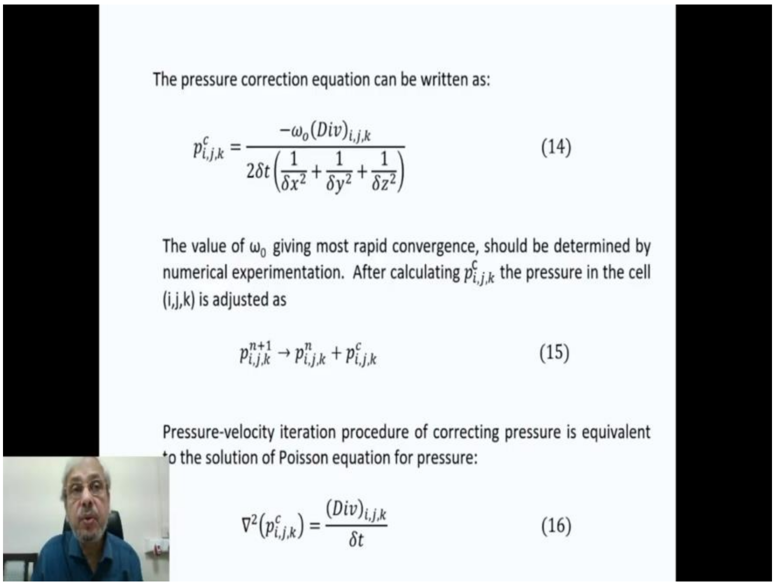
$$\left[\frac{\tilde{u}_{i,j,k}^{n+1} - \tilde{u}_{i-1,j,k}^{n+1}}{\delta x} + \frac{\tilde{v}_{i,j,k}^{n+1} - \tilde{v}_{i-1,j,k}^{n+1}}{\delta y} + \frac{\widetilde{w}_{i,j,k}^{n+1} - \widetilde{w}_{i-1,j,k}^{n+1}}{\delta z}\right]$$

And definitely, this (Right hand side of Eq. 13 in Slide Time: 11:04) is now non-zero because if this is zero, if all the terms equal to zero, then continuity has been satisfied, but since we have used provisionally advance quantity, these(Right hand side of Eq. 13 in Slide Time: 11:04) bracketed terms will produce non-zero terms and then, there will be pressure terms and if you look at the pressure terms, these are pressure corrections at i, j, k cell $i+1, j, k \quad i-1, j, k \quad then\ i, j+1, k \quad i, j-1, k \quad i, j, k+1 \quad i, j, k-1$.

So, i, j, k and all its you know six neighbors $i+1, j, k \quad i-1, j, k \quad i, j+1, k \quad i, j-1, k \quad i, j, k+1 \quad i, j, k-1$ those all-pressure corrections terms will come here. Now, you see we have written a special attribute that, neglecting pressure corrections in the neighboring cells. So, we just retain pressure when we sum up this, pressure corrections at i, j, k cell.

So, pressure correction at $i+1, j, k \quad i-1, j, k \quad i, j+1, k \quad i, j-1, k \quad i, j, k+1 \quad i, j, k-1$, we are neglecting and definitely, this is you know this may not be proper to neglect that and for that, we have to really pay some price, we will see that also. So, by neglecting, we can write this term which is equation 13 and from equation 13, then deliberately we neglected, you can also sense we why we neglected to get rid of implicitness so that we get an explicit expression for pressure correction.

(Refer Slide Time: 21:10)



The pressure correction equation can be written as:

$$p^c_{i,j,k} = \frac{-\omega_o (Div)_{i,j,k}}{2\delta t\left(\frac{1}{\delta x^2}+\frac{1}{\delta y^2}+\frac{1}{\delta z^2}\right)} \qquad (14)$$

The value of ω_0 giving most rapid convergence, should be determined by numerical experimentation. After calculating $p^c_{i,j,k}$ the pressure in the cell (i,j,k) is adjusted as

$$p^{n+1}_{i,j,k} \rightarrow p^n_{i,j,k} + p^c_{i,j,k} \qquad (15)$$

Pressure-velocity iteration procedure of correcting pressure is equivalent to the solution of Poisson equation for pressure:

$$\nabla^2\left(p^c_{i,j,k}\right) = \frac{(Div)_{i,j,k}}{\delta t} \qquad (16)$$

So, from equation 13, we get pressure correction at i, j, k I am going back again at i, j, k you can see. then if this (left hand side of equation 13) is 0, ideally this is 0 so, then the and this (left hand side of equation 13) is non-zero so, this provisionally advance velocity what they will produce any cell, we will get that as non-zero divergence. So, non-zero divergence divided by the coefficients of $p_{i,j,k}$ term.

So, basically, then $p_{i,j,k}$ correction is divergence at each cell, this divergence is basically arising out of the provisionally advanced velocity components (refer Eq. 14). These (denominator of Eq.14) were basically the coefficients of $p_{i,j,k}$ term, we are clubbing them together and transferring on the right-hand side.

And we are supposed to get pressure correction through this, but we are advancing the, accelerating our level of pressure correction, we are using level over relaxation term, the value ω_0, this is over relaxation factor giving most rapid convergence should be determined through numerical experiments. Numerically, one can there are ways to experiment with it. Usually, the value which we use I can give you the value it lies between 1.5 and 1.7.

But you know one can do also testing, a priori testing to a choose a correct over relaxation factor or you know I should not say correct, more appropriate over relaxation factor. So, after calculating $p^c_{i,j,k}$ at each cell, we can correct the pressure. So, pressure at n^{th} level, we add pressure correction and we are supposed to get $p^{n+1}_{i,j,k}$ at the level $n+1$.

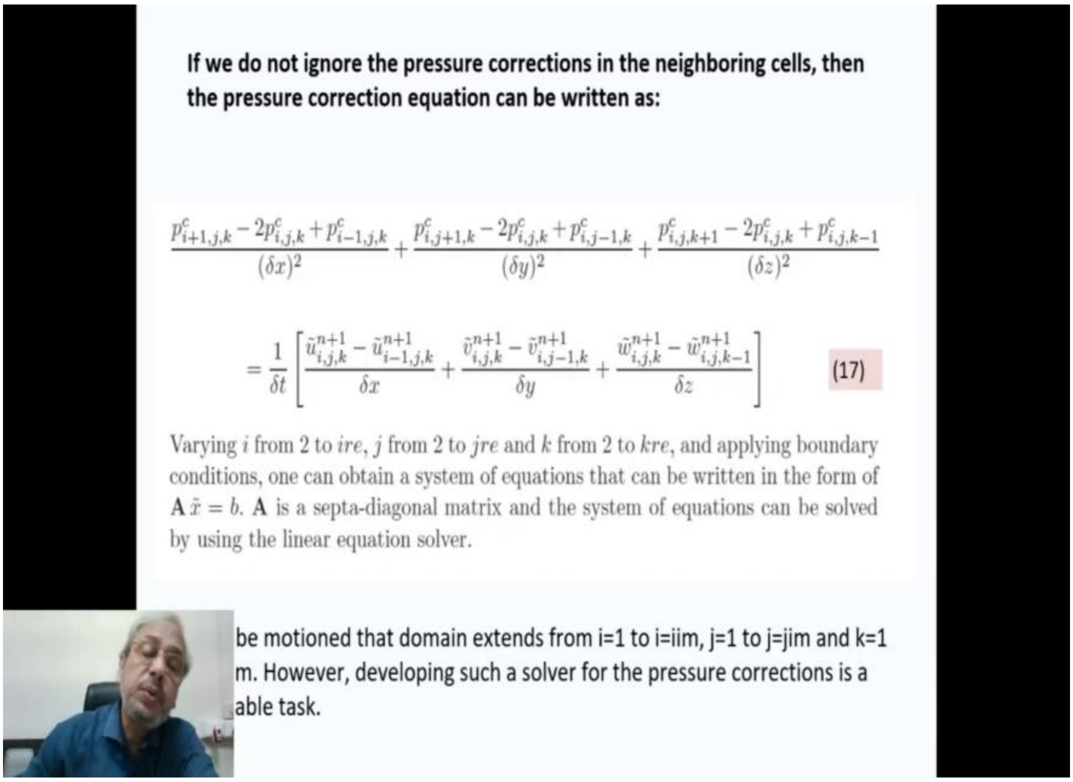
But see you can jolly well understand that we neglected all the neighboring terms to get this explicit expression. So, you cannot expect that this pressure correction is really the exactly correct final pressure correction term and that is why, here (Eq. 15 slide time 21:10) also used instead of equality sign, tends to sign.

So, pressure has to be corrected and after having the pressure correction, we add it with n^{th} level pressure, we get improved $p_{i,j,k}$ but that is not final $p_{i,j,k}$ and we repeat this calculation what we did and maybe some finite time of I mean setting some loop and maybe calculating it iteratively, it may take you know 100 iterations it may take 200 iterations or whatever the number of iterations.

But finally, when the correct velocity quantities evolve, divergence will be 0, there will be no further pressure correction, we get the correct pressure and by that time, all the velocity terms have also evolved as correct velocities from provisional to corrected final velocity.

So, this is basically called pressure velocity iteration. Now, this pressure velocity iteration procedure if you really look at it, it is basically the way we are solving pressure correction, we can call this as a solution of Poisson's equation for pressure and this can be shown also. Just I will show you in the next slide.

(Refer Slide Time: 26:02)



If we do not ignore the pressure corrections in the neighboring cells, then the pressure correction equation can be written as:

$$\frac{p^c_{i+1,j,k} - 2p^c_{i,j,k} + p^c_{i-1,j,k}}{(\delta x)^2} + \frac{p^c_{i,j+1,k} - 2p^c_{i,j,k} + p^c_{i,j-1,k}}{(\delta y)^2} + \frac{p^c_{i,j,k+1} - 2p^c_{i,j,k} + p^c_{i,j,k-1}}{(\delta z)^2}$$

$$= \frac{1}{\delta t}\left[\frac{\tilde{u}^{n+1}_{i,j,k} - \tilde{u}^{n+1}_{i-1,j,k}}{\delta x} + \frac{\tilde{v}^{n+1}_{i,j,k} - \tilde{v}^{n+1}_{i,j-1,k}}{\delta y} + \frac{\tilde{w}^{n+1}_{i,j,k} - \tilde{w}^{n+1}_{i,j,k-1}}{\delta z}\right] \quad (17)$$

Varying i from 2 to ire, j from 2 to jre and k from 2 to kre, and applying boundary conditions, one can obtain a system of equations that can be written in the form of $\mathbf{A}\,\tilde{x} = b$. $\mathbf{A}$ is a septa-diagonal matrix and the system of equations can be solved by using the linear equation solver.

be motioned that domain extends from i=1 to i=iim, j=1 to j=jim and k=1
m. However, developing such a solver for the pressure corrections is a
able task.

If we do not ignore the pressure corrections in the neighboring cells, then the pressure correction equation may be written as:

$$\frac{p^c_{i+1,j,k} - 2p^c_{i,j,k} + p^c_{i-1,j,k}}{(\delta x)^2} + \frac{p^c_{i,j+1,k} - 2p^c_{i,j,k} + p^c_{i,j-1,k}}{(\delta y)^2} + \frac{p^c_{i,j,k+1} - 2p^c_{i,j,k} + p^c_{i,j,k-1}}{(\delta z)^2}$$
$$= \frac{1}{\delta t}\left[\frac{\tilde{u}^{n+1}_{i,j,k} - \tilde{u}^{n+1}_{i-1,j,k}}{\delta x} + \frac{\tilde{u}^{n+1}_{i,j,k} - \tilde{u}^{n+1}_{i,j-1,k}}{\delta y} + \frac{\tilde{u}^{n+1}_{i,j,k} - \tilde{u}^{n+1}_{i,j,k-1}}{\delta z}\right]$$

All these correction terms if those are retained together with $p^c_{i,j,k}$ then we will get you know left-hand side of the equation like this and right-hand side will remain the same (in above equation). Basically, this is you know level the continuity equation written in terms of provisional level provisionally advanced velocity components. And then, you can see that we cannot directly get an explicit expression for pressure correction at i, j, k point.

Earlier, we got an explicit expression for $p^c_{i,j,k}$ but here we cannot get that, all the neighbors are a present correction pressure correction term. So, what we have to do? We have to vary i from 2 to ire, ire means the maximum number of cells or maximum number of points something like $i_{max} - 1$. So, here of course, the domain is i is 1 to iim. So, this is iim minus 1.

So, because the extreme points like iim on 1 extreme and 1 at the other extreme these are to be taken care of by the boundary conditions. So, varying i from 2 to ire, j from 2 to jre and k from 2 to kre and applying boundary conditions, we can obtain a system of equations that can be written in the form $A\tilde{x} = b$ that is A is a matrix, $\tilde{x}$ is unknown vector and equal to b, b is a right-hand side known vector.

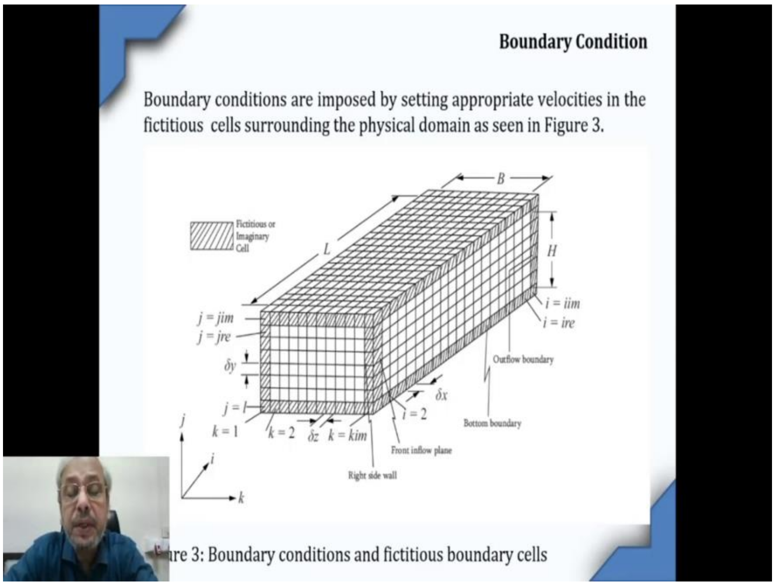
A is a septa-diagonal matrix and system of equation can be solved by setting up a linear solver, linear equation solver.

Today, quite a few advanced open-source linear solvers are available like you know by cgs tab or hypri etcetera those can be linked also, but at the moment, you know we will not to go or we will not take up our discussion to that direction, what we can say that basically, you know one can solve the pressure correction equation also this way by setting up a matrix and having the pressure correction terms solved directly from the matrix.

But as you know, setting up such you know metrics equation and solution techniques, both are very much involved that is why you know the first option that we discussed that explicit form of pressure correction equation. Although, we are ignoring neighboring points that is why you need to iterate you know the process or the pressure correction and corresponding velocity correction in a number of times iterations can go to you know 200, 300 times I mean continue for 200, 300 you know sort of iterations.

But you know solution technique is straightforward and simple. Here also, you can see we one if you know one desires to go for this way that this can also be done, but developing such a solver is quite involved and you know requires special effort. I have also mentioned that the domain extends from i equal to 1 to iim, j equal to 1 to jim and k equal to 1 to kim. These are the maximum number of points in respective directions.

(Refer Slide Time: 32:54)



Boundary Condition

Boundary conditions are imposed by setting appropriate velocities in the fictitious cells surrounding the physical domain as seen in Figure 3.

re 3: Boundary conditions and fictitious boundary cells

Now, we will discuss about the boundary conditions. Boundary conditions are imposed by setting appropriate velocities in the fictitious cells surrounding the physical domain as seen in the figure. So, what we can see in the figure that basically, on each side, we have fictitious cells and these fictitious cells, they participate in computation because we get our boundary defined by setting up the velocity values in the fictitious cells.

In such a way that you know all the desirable conditions for impervious boundary or no slip boundary or free slip boundary, those are properly described with the help of fictitious cells. But when we do the post processing obviously, we need not pay attention to the velocity values to the in the fictitious cells because that is outside the physical domain.

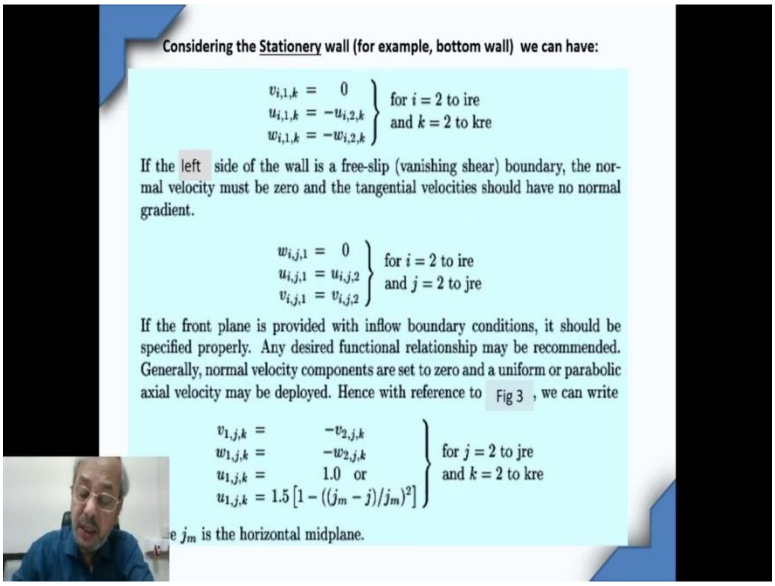
So, fictitious cells are also called imaginary cells and as you can see that they are outside the physical domain like you know in x-direction for example, the physical domain extends for i equal to 2 to i equal to ire, but iim and i equal to 1, these cells are all imaginary cells.

Similarly, in j direction, j equal to 2 to jre, this extent you know these cells cover the physical domain, but we have nomenclature wise j equal to 1 to jim. So, all these cells at jim level, all the cell at j equal to 1 level, these are fictitious cells and these are used for as I have said that setting up correct boundary conditions for the velocities at the planes where velocities are not directly defined.

So, what we can conclude from this slide that in x-direction, we have i equal to 1 to iim cells, in y-direction, we have j equal to 1 to jim cells, in z-direction, we have k equal to 1 to kim number of cells and cell width in x-direction δx, in y direction δy, in z direction δz and you know the real physical domain when we try to identify physical domain will exclude imaginary cells.

But imaginary cells are very important because they participate in computation to set up the correct boundary conditions at the confining walls because many times for staggered rate, you will not get velocity level components are directly defined on the planes of interest.

(Refer Slide Time: 36:57)



Considering the Stationery wall (for example, bottom wall) we can have:

$$\left.\begin{aligned} v_{i,1,k} &= 0 \\ u_{i,1,k} &= -u_{i,2,k} \\ w_{i,1,k} &= -w_{i,2,k} \end{aligned}\right\} \text{for } i = 2 \text{ to ire and } k = 2 \text{ to kre}$$

If the left side of the wall is a free-slip (vanishing shear) boundary, the normal velocity must be zero and the tangential velocities should have no normal gradient.

$$\left.\begin{aligned} w_{i,j,1} &= 0 \\ u_{i,j,1} &= u_{i,j,2} \\ v_{i,j,1} &= v_{i,j,2} \end{aligned}\right\} \text{for } i = 2 \text{ to ire and } j = 2 \text{ to jre}$$

If the front plane is provided with inflow boundary conditions, it should be specified properly. Any desired functional relationship may be recommended. Generally, normal velocity components are set to zero and a uniform or parabolic axial velocity may be deployed. Hence with reference to Fig 3 , we can write

$$\left.\begin{aligned} v_{1,j,k} &= -v_{2,j,k} \\ w_{1,j,k} &= -w_{2,j,k} \\ u_{1,j,k} &= 1.0 \quad \text{or} \\ u_{1,j,k} &= 1.5\left[1-((j_m-j)/j_m)^2\right] \end{aligned}\right\} \text{for } j = 2 \text{ to jre and } k = 2 \text{ to kre}$$

e j_m is the horizontal midplane.

And immediate example is stationary bottom wall. So, if I will go back by one slide. Bottom wall that means, if you exclude imaginary cell and the domain where I mean physical domain described by this all these internal cells and you know the domain boundary that means, stationary bottom wall for example, this is if we exclude one layer of imaginary cells, just you know the top of that layer it is the bottom boundary exist. But since bottom boundary exists on the top of bottom most imaginary cell layer so, we have to see how we can set up the boundary conditions so that the rigid no slip bottom wall can be defined. We have defined $v_{i,1,k}$ this is a normal component of velocity and i equal to 2 to ire, k equal to 2 to kre we will vary.

And all v velocities if you imagine the bottom most layer of cells, all v velocities are falling directly on the bottom plane so, we can set them equal to 0. But $u_{i,1,k}$ these cells are defined at the middle of you know the bottom most cells not following on the boundary. So, the velocities at these cells $u_{i,1,k}$ are to be interpolated with $u_{i,2,k}$ in such a way that the bottom wall can be defined or on the bottom wall u velocity satisfies the no slip boundary condition, u equal to 0.

This is impervious boundary condition, normal velocity equal to 0 and u and v, u normal velocity equal to 0, $v_{i,1,k}$ equal to 0 and u and w, these are tangential components of velocity. So, these are to be interpolated with the immediate upper layer of cells in such a way that bottom boundary is created as no slip boundary.

So, $u_{i,1,k} = -u_{i,2,k}$, $w_{i,1,k} = -w_{i,2,k}$, i equal to 2 to ire, k equal to 2 to kre. Now, this is how we set the boundary velocity boundary condition in such a way that immediate at the top of this layer of imaginary cells, we define a region no slip wall.

Similarly, for example, if we want to define this plane which is you know on the left-hand side, if we exclude imaginary cell, the plane which we will get on this domain, if we want to define that plane as free slip wall then what we have to do? We have to setup the boundary conditions in the following way that the normal velocity must be 0 on that plane and tangential velocity should have no normal gradient.

So, that means, normal to this plane is basically w component of velocity, we can directly set those velocities to 0 and, but we have to interpolate u and v component in such a way that physically, this plane becomes a free slip plane which is describable by $\frac{\partial u}{\partial z}$ and $\frac{\partial v}{\partial z}$ equal to 0 that means, $u_{i,j,1} = u_{i,j,2}$ i equal to 2 to ire, j equal to 2 to jre, we will describe the plane.

Similarly, $v_{i,j,1} = v_{i,j,2}$; $\frac{\partial v}{\partial z} = 0$, $\frac{\partial u}{\partial z} = 0$ is deciding u velocity boundary conditions and $\frac{\partial v}{\partial z} = 0$, we will dictate basically boundary condition on v velocity on the plane i equal to 2 to ire j equal to 2 to jre and k equal to 1. All these boundary conditions are first set of boundary conditions is for rigid no slip wall, a second set of boundary conditions are fall you know free slip wall or vanishing share boundary condition.

Now, on the front plane for example, this is there is no fixed prescription for that front plane, you can, or we can set up, we can define the velocities the way we like, it can be experimentally obtained velocity profile, it may be uniform velocity profile, it may be parabolic velocity profile. So, for I mean, and we have to setup a function on you knows basically this is j-k plane and in the i direction, we have to set up a function to describe the proper profile.

Like, if we want to set up a parabolic velocity profile, what we have done? v velocity we have set to 0. Basically, you can see $v_{1,j,k}$ equal to minus $v_{2,j,k}$. Here j equal to 2 to jre and k equal to 2 to kre on any i plane basically, we will vary j 1 to j extreme j 2 to k 1 to k extreme that will you know describe any plane in i direction.

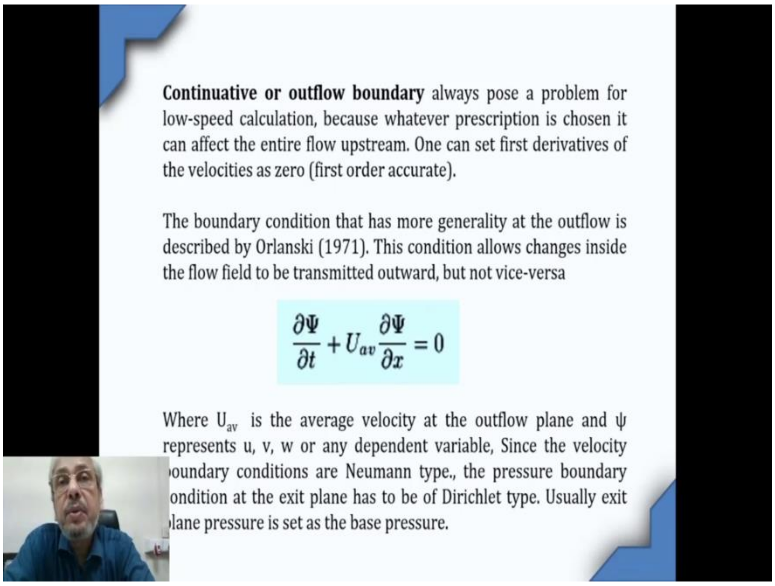
So, at the entry i equal to 1, but j will vary from you know 1 to j max or jim, k will vary from 1 to kim, but one need not be included in setting up the algorithm because if we set k equal to 2, the boundary condition will be inducted with that.

So, $v_{1,j,k} = -v_{2,j,k}$, $w_{1,j,k} = -w_{2,j,k}$ and so, w component and v component velocity, we are setting on that plane as 0 and $u_{1,j,k}$ equal to 1. This is basically uniform velocity profile and $u_1, j, k = \frac{3}{2}\left[1 - \left(\frac{j_m - j}{j_m}\right)^2\right]$ this is basically y_m is the horizontal mid plane. So, basically setting up, we are trying to set up a parabolic velocity profile.

So, $v_{1,j,k} = -v_{2,j,k}$, makes v on this plane, inflow plane 0, $w_{1,j,k} = -w_{2,j,k}$ also make w velocity 0 and u either 1 or parabolic or any other profile, we can set up by describing a mathematical function representing the profile and j equal to 2 to jre and k equal to 2 to kre that is basically any cross plane will be covered by this loops or this variations j equal to bottom to top extent k equal to one side to another side, will give the cross plane.

So, that is how we have, we can set inflow plane in flow boundary condition on the inflow plane, inflow plane exist here, one can understand by looking at the geometry.

(Refer Slide Time: 48:24)



Continuative or outflow boundary always pose a problem for low-speed calculation, because whatever prescription is chosen it can affect the entire flow upstream. One can set first derivatives of the velocities as zero (first order accurate).

The boundary condition that has more generality at the outflow is described by Orlanski (1971). This condition allows changes inside the flow field to be transmitted outward, but not vice-versa

$$\frac{\partial \Psi}{\partial t} + U_{av}\frac{\partial \Psi}{\partial x} = 0$$

Where U_{av} is the average velocity at the outflow plane and ψ represents u, v, w or any dependent variable, Since the velocity oundary conditions are Neumann type., the pressure boundary ondition at the exit plane has to be of Dirichlet type. Usually exit lane pressure is set as the base pressure.

Now, we will go for continuity or outflow boundary. Now, continuity or outflow boundary, if we have to set up outflow boundary so, basically, we have to give boundary conditions here right. If we exclude one layer of imaginary cells at the extreme side of the geometry, extreme downstream of the geometry so, basically, we have to define exit boundary here.

Now, there is a catch with the exit boundary because exit boundary condition is not known, had we know known the exit plane velocities, I mean we would not have solved this governing equations, these are not known, but since we are you know solving a problem which is elliptic in space and parabolic in time, we have to prescribe boundary conditions for all confining surfaces.

Out of this compulsion, we have to setup some functions at the outflow plane which depicts correct flow physics. Now, one can you know very easily imagine a set of conditions like you know a first derivatives of velocities as zero in the flow direction. So, basically x-direction is the flow direction. So, , $\frac{\partial u}{\partial x} = 0$, $\frac{\partial v}{\partial x} = 0$, $\frac{\partial w}{\partial x} = 0$ that is a plausible boundary condition.

But this boundary conditions if are first order accurate, but you know not wrong either only thing accuracy can be enhanced. If you look at physically, $\frac{\partial u}{\partial x} = 0$, $\frac{\partial v}{\partial x} = 0$, $\frac{\partial w}{\partial x} = 0$, these boundary conditions also in a way conveying the message that acceleration is 0, spatial acceleration is 0 which is one firm of restriction.

Actually, you know it will be better if we do not impose any such restriction, outflow boundary should be a free boundary. Physically, the flow should have a smooth transition or smooth transition from the flow domain, smoother exit from the flow domain.

So, the boundary condition that has more generality at the outflow plane is described by Orlanski, this was a very well-known way I mean well referenced also paper in journal of this was published in journal of computational physics and these conditions are called Orlanski condition which is basically $\frac{\partial\psi}{\partial t}+\frac{U_{av}\partial\psi}{\partial x}=0$, where ψ stands for the dependent variables, u, v, w, t all the level if you have energy equation it is also imposed on t temperature.

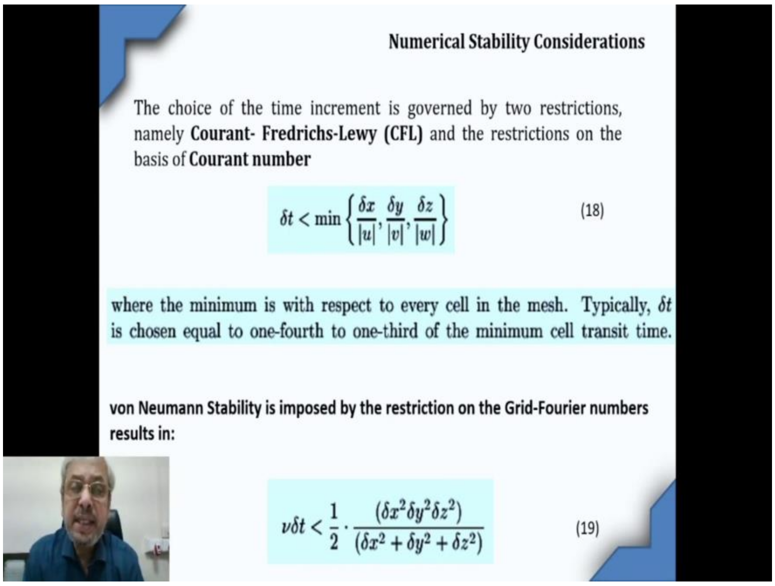
So,u, v, w, t you know ψ represents all those variables and $\frac{\partial\psi}{\partial t}+\frac{U_{av}\partial\psi}{\partial x}=0$ and this U_{av} is the basically average velocity at the exit plane and obviously, when we are computing for a given time step, it is not known at that time-steps so, it should be known from the previous time step.

So, from the previous time step value or values, U_{av} has to be calculated and that U_{av} has to be prescribed here and then, we will be able to set up a very correct boundary condition on u, v, w and if you have temperature also energy equation when you are competing on t also.

Now, you can see if you use simply, you know first derivative equal to 0 or if you use Orlanski condition, whatever we do, we are basically setting up boundary conditions that are Neumann type. So, if the velocity boundary conditions are Neumann type, the pressure boundary condition has to be Dirichlet type and indeed, we do that what we do? We set up the pressure boundary condition or pressure at the exit cells as the base pressure, base pressure of the domain.

As we know that every iteration pressure is being corrected so, through pressure correction, all the upstream cell pressures are corrected in order to set up a correct pressure gradient in the domain, but exit plane pressure remains as the base pressure, we do not correct exit plane pressure.

(Refer Slide Time: 55:10)



Numerical Stability Considerations

The choice of the time increment is governed by two restrictions, namely **Courant- Fredrichs-Lewy (CFL)** and the restrictions on the basis of **Courant number**

$$\delta t < \min\left\{\frac{\delta x}{|u|}, \frac{\delta y}{|v|}, \frac{\delta z}{|w|}\right\} \qquad (18)$$

where the minimum is with respect to every cell in the mesh. Typically, δt is chosen equal to one-fourth to one-third of the minimum cell transit time.

von Neumann Stability is imposed by the restriction on the Grid-Fourier numbers results in:

$$\nu\delta t < \frac{1}{2}\cdot\frac{(\delta x^2\delta y^2\delta z^2)}{(\delta x^2+\delta y^2+\delta z^2)} \qquad (19)$$

Having said this so, we have covered a basic flow I mean solution strategy, boundary conditions among all related pressure velocity iteration now, we can see that since our governing equation is elliptic in space, parabolic in time so, we will basically march in time direction. So, the time step will have stability requirement and if you recall for such equations where it is elliptic in space and parabolic in time, we pose both the restrictions on the time step.

One is basically restriction which is known as Courant-Fredrich-Lewy condition or the restriction on Courant number. Now, that means, at each cell, Courant number should be less than 1 and local Courant number at each cell is defined by for example, in x-direction, $\frac{u\delta t}{\delta x}$, in y-direction, $\frac{v\delta t}{\delta y}$ and in z-direction, $\frac{w\delta t}{\delta z}$.

So, delta t minimum over the domain, we can write this way δx by modulus of u, u may be locally negative or positive, δy by modulus of v, δz by modulus of w, where the maximum is with respect to every cell in the mesh and minimum is also with respect to every cell in the mesh.

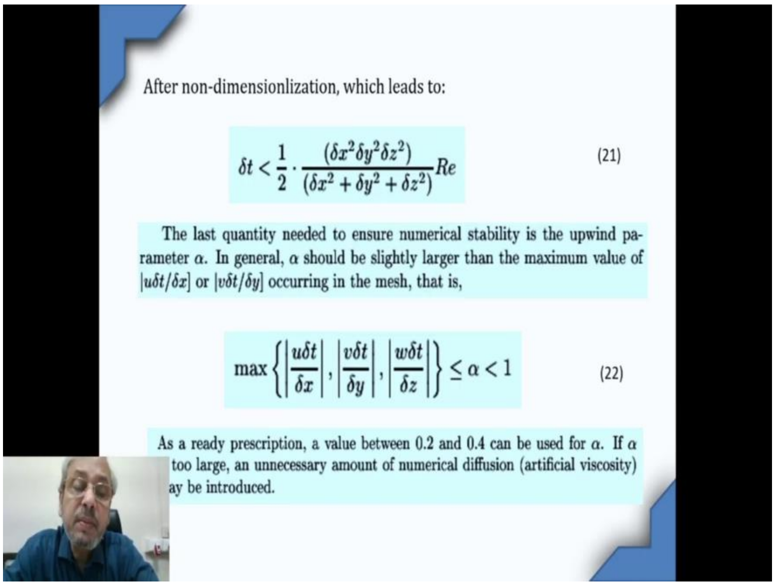
So, basically, this or you know in each cell, whatever is the Courant number corresponding you know the cell velocity at that cell from there, we can calculate δt and here, minimum over the entire domain, we calculate δt. Typically, delta t is chosen equal to finally, see this minimum I mean less than minimum of this Courant I mean δt at all cells based on

Courant number in x direction, y direction and z direction that is well, but and it is less than minimum of that, but how much less that also one has to know or estimate.

So, typically delta t is chosen equal to one-fourth or one-third of the minimum cell transit time. So, delta t in each cell is calculated, from that minimum is found out and whatever is that value usually you know one-third or one-fourth of that value is used for time advancement that is from one side.

And then, we can apply Neumann stability condition also and one Neumann stability restriction means it is on grid Fourier number that means, it is on you know the coefficient in this case, $\nu\delta t/\delta x^2$ or $\nu\delta t/\delta y^2$, $\nu\delta t/\delta z^2$ square and a $\nu\delta t$ will be minimum of all these and this can be summarized as equation 19 (refer Slide Time: 55:10).

(Refer Slide Time: 59:42)



After non-dimensionlization, which leads to:

$$\delta t < \frac{1}{2}\cdot\frac{(\delta x^2\delta y^2\delta z^2)}{(\delta x^2+\delta y^2+\delta z^2)}Re \qquad (21)$$

The last quantity needed to ensure numerical stability is the upwind parameter α. In general, α should be slightly larger than the maximum value of $|u\delta t/\delta x|$ or $|v\delta t/\delta y|$ occurring in the mesh, that is,

$$\max\left\{\left|\frac{u\delta t}{\delta x}\right|,\left|\frac{v\delta t}{\delta y}\right|,\left|\frac{w\delta t}{\delta z}\right|\right\}\le\alpha<1 \qquad (22)$$

As a ready prescription, a value between 0.2 and 0.4 can be used for α. If α too large, an unnecessary amount of numerical diffusion (artificial viscosity) ay be introduced.

From here, we can if we non-dimensionalize, then we will get $\delta t < \frac{1}{2}.\frac{\delta x^2\delta y^2\delta z^2}{(\delta x^2+\delta y^2+\delta z^2)}Re$. So, δt is less than this (right side of 21) quantity given by equation 21. So, now, equation 21 gives indication of δt, also equation 18 gives indication of δt. We calculate basically, you know both the δts in the program and use minimum of these two effectively finally, use minimum of this two. So, whatever δt we get with for Neumann stability analysis and whatever δt we get using Courant number or Courant-Fredrich-Lewy CFL condition, we get δt which is you know minimum of these two δts.

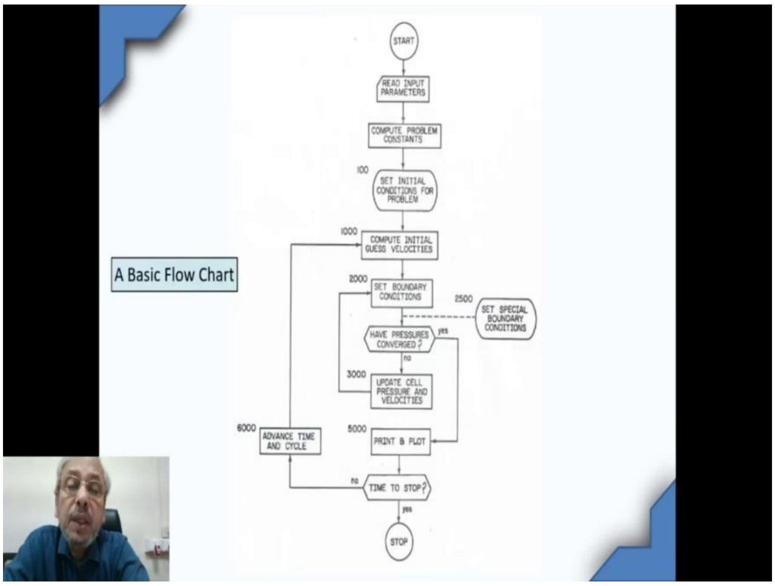
The last quantity needed to ensure numerical stability is the upwind parameter α. You recall that in the discretization, we defined this α which is contribution of basically upwinding and this upwind parameter is α.

In general, α should be slightly larger than the maximum value of $u\delta t/\delta x$, $v\delta t/\delta y$ occurring in the mesh that is maximum of $u\delta t/\delta x$, $v\delta t/\delta y$, $w\delta t/\delta z$ over the whole domain you know whatever value we get, α is slightly larger than that, but less than 1.

As a ready prescription, a value between 0.2 and 0.4 can be used for α. If α is too large, an unnecessary amount of numerical diffusion or artificial viscosity will be introduced. So, α is needed for stable calculation that we know, but this α value should be minimum so that the formulation tends towards second order accuracy.

So, for that alpha 0.2 or 0.3 or even we can use 0.1 if there is no problem related to stability. So, that should be experimented, numerical experiment can be done that you know how small α we can take so that formal accuracy or over all accuracy of the; accuracy of the discretized equations remain second order spatially.

(Refer Slide Time: 63:17)



A Basic Flow Chart

START
READ INPUT PARAMETERS
COMPUTE PROBLEM CONSTANTS
100
SET INITIAL CONDITIONS FOR PROBLEM
1000
COMPUTE INITIAL GUESS VELOCITIES
2000
SET BOUNDARY CONDITIONS
2500
SET SPECIAL BOUNDARY CONDITIONS
HAVE PRESSURES CONVERGED ?
yes
no
3000
UPDATE CELL PRESSURE AND VELOCITIES
6000
ADVANCE TIME AND CYCLE
5000
PRINT & PLOT
no
TIME TO STOP?
yes
STOP

Now, as you can see, I have given a basic flow chart start, read input parameters, compute problem constants, set initial conditions for the problem, compute initial guess velocities. Now, at 2000 level, these levels have been given numbers that means, these levels are basically signifying different sections of the code, different sessions of the program, set boundary conditions. If you have additional boundary condition that means, we have only discussed confining boundary conditions, this may be a flow where you have many obstacles inside maybe you know flow first circular cylinder, flow first a sphere, flow first a square cylinder, flow first any other object within the computational domain. So, set of those special boundary condition to describe the obstacle.

Then, have the pressure converged. If it is converged find, go out. If it is not converged, then update cell pressures and velocity. This is so called pressure velocity iteration which we have already discussed you know at the 3000 level. So, this loop is set, it will keep on iterating and you know; it will ask whether pressure has converged. If pressure is converged, it will go out of the iteration loop, if it has not converged, it will again enter the loop.

If the pressure is converged that means, at a level, if we have got correct pressure at each cell, correct velocities at each cell that means, divergence field velocity field, we have gotten the solution of velocity and pressure for that time level. So, we can then print and plot, but if that is not the right physical situation that means, if it is; if it is unsteady flow, then of course, you know it will keep on continuing, but if it is a steady flow and we need some more iterations to reach the steady state condition, then outer loop again, we will activate.

So, advance the time cycle. So, one is outer loop to advance in the time cycle from n to $n + 1$, to $n + 2$ to $n + 3$ and another is inner loop for a given time step or a given you know level, we have to get converged velocity and pressure field. So, this is the overall flow chart.

And then, depending on the requirement of the problem, this can be modified like you know this time to stop this loop you know either for you go for a you know steady state computation or maybe you go for some desire level of unsteady flow and all those things can be done.

And all intermediate unsteady flows if we want to store, we will have to make use of this level, 5000 level, here it is written print and plot so, that means, you know we can print the values, we can plot the values or we need not even you know do either of these, we can simply create a data file, we can store the data in a file and come out.

So, that is the loop for computation, and this is a basic flow chart. Thank you, we will stop here today. In a next class, we will extend our analysis little further. We will take up energy equation and we will try to give examples of possible flow fields and its associated energy equation and temperature field.

Thank you very much.