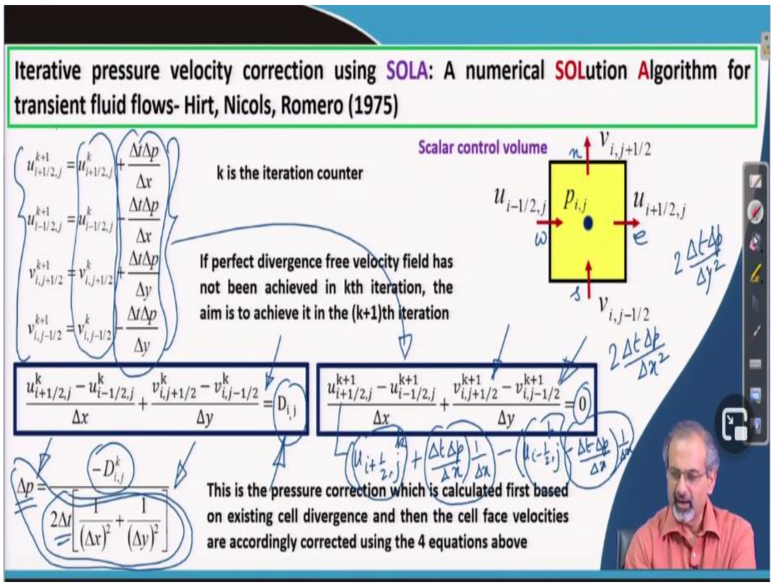
Introduction to CFD
Prof. Arnab Roy
Department of Aerospace Engineering
Indian Institute of Technology - Kharagpur

Lecture - 44
Numerical Solution of Two Dimensional Incompressible Navier Stokes Equations (continued)

(Refer Slide Time: 00:30)



In this lecture, we will complete our discussion on two dimensional incompressible Navier Stokes equations. So, in the previous lecture, we have started the discussion on the iterative pressure velocity correction using the SOLA algorithm. And we had already discussed on how the new component of velocities on the east and west faces of the scalar control volume are corrected.

Similar strategies are used for calculating the k + 1th iteration level values of v at the north and south faces of the control volume. So, in this case, of course, if you watch carefully, it depends on delta p by delta y in a positive and a negative sense. Now, the point is we still do not know what this delta p is all about. Therefore, as long as we do not have any clue about this delta p, we cannot take the velocities to the next iteration level.

So, we have to find a way a means by which we define this delta p. And of course, it has to be defined in a way such that we are able to move closer to attaining zero dilatation for each and every cell this way. So, let us see how we do it in SOLA. So, once we substitute these equations that we have derived for the k + 1th iteration level, we understand that they should ideally satisfy zero divergence that is at least the target.

And if we look at the kth iteration level, then we still have some nonzero divergence over here. Now, this D this capital D may be a positive maybe a negative value irrespective of what the sign is, we have to limit it towards 0. If it is a positive value that means that as though some flow is actually leaking out of the cell into the surroundings. And therefore, we must be having more than optimum pressure in that cell which is making the flow leak out of the cell, which means the pressure has to be corrected in such a manner that the velocities come down in that cell.

And then we are able to reach zero dilatation. It is just the opposite if the dilatation is negative. That means you have lower than optimum pressure in the cell. That is why flow is getting an opportunity to leak into the cell from the neighborhood. So, you need to raise the pressure to oppose that leakage into the cell so, that it gets balanced out. Remember the job is complicated in this sense that if you do it individually to one cell, it does not solve the problem.

It has to be done in unison for all the cells which are there in the domain. That means there is a global balancing act which has to go on till you reach a kind of negligible extent of dilatation that means negligible extent of mass defect in the entire system of cells which make your domain, right. So, the main crux of the problem is to find an expression for delta p. And how do we do it? We just do it by substituting these equations into this requirement that the k + 1th iteration values should satisfy your zero divergence.

If you substitute it there, then all these u’s and v terms in the kth time level will generate this divergence. And the remaining terms that mean, these terms will generate an expression in terms of delta p times some coefficient. And what is that coefficient? That happens to be this. Alright, so, that is how we are able to come to this expression. So, I am just trying to repeat it once more.

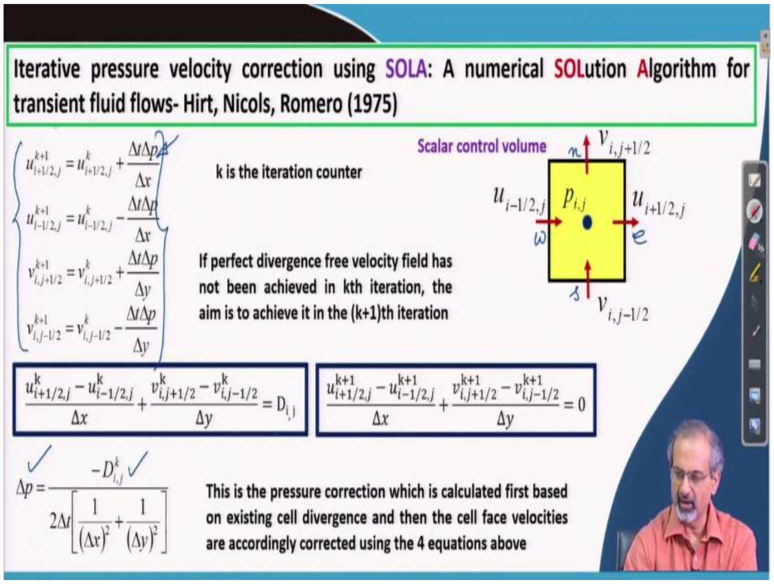
That is we substitute these equations into the equation for the discrete continuity equation into the discrete continuity equation for the k + 1th iteration level where we are trying to attain a zero divergence or dilatation. If you do that, you can understand here you will get a u i plus half j at kth iteration plus delta t delta p by delta x the whole divided by delta x. This is what the first term will generate.

What will be the second term generating? The second term will generate minus u i minus half j k minus delta t delta p by delta x the whole divided by delta x. And it will also go on for the v terms. Now, what are we going to get out of it? From these terms, we are going to get the contribution to build that term D. And what will these terms generate? Already it has generated delta t delta p by delta x square two times for u.

So, likewise, you will get a two delta t delta p by delta y square if you account for the v terms. So, what do you have finally? You have the same thing figuring out here if you watch carefully. So, it is two times delta t delta p into one by delta x square plus 1 by delta y square. So, we have therefore, generated an equation for delta p, in terms of what? In terms of the dilatation of the cell the time step and the geometric details of the cell.

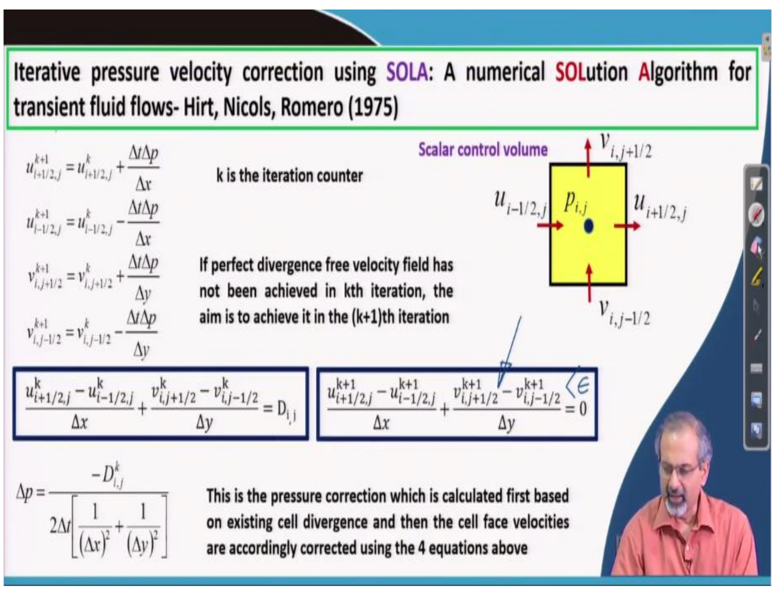
That means the grid spacing delta x and delta y. So, now, the idea is to feed back this value of delta p into those four equations and that is how you can actually take your velocities to the next iteration level, because now you have the value of delta p. For each and every cell with the current iteration levels, you can generate your dilatation.

(Refer Slide Time: 07:22)



That means D is known to you, once D is known to you, delta p is known to you, if delta p is known to you, you can substitute it here in all these four equations. If you can do that, you can generate all these updates. And the idea is you need to check that with the updates how close have you reached to the zero dilatation condition.

(Refer Slide Time: 07:49)

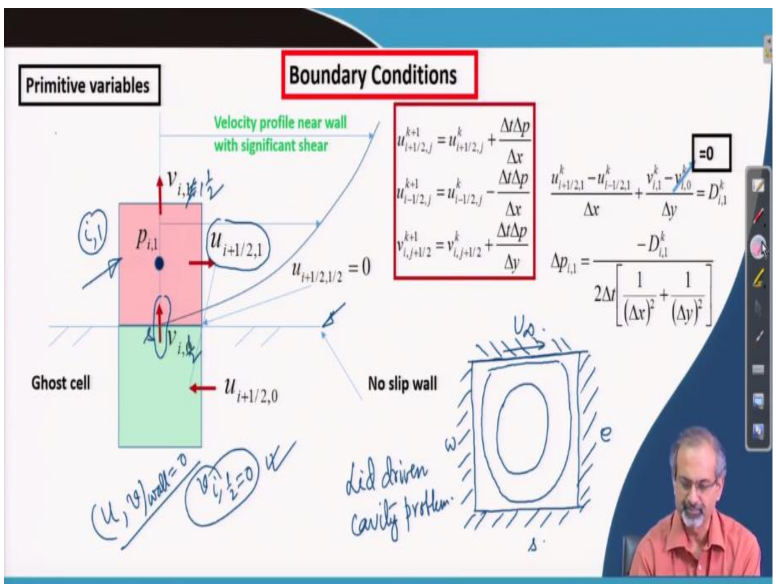


So, the check that you have to ensure is that have you really reached this condition. If not zero have you reached a number which is less than epsilon on the right hand side? If it is true for all the i’s and j’s that comprise the domain, then you stop the iteration there. So, this is how the SOLA algorithm works, where pressure and velocity gets iteratively defined till you reach a zero divergence condition.

So, we have learned another manner another way in which pressure velocity coupling can work in an iteratively corrected manner so, that we finally achieve a zero dilatation field. So, we are now sufficiently equipped to handle this problem of pressure velocity coupling in incompressible Navier Stokes equations involving primitive variables. And we have learnt it in two dimensional sense.

But as you can understand it should be a routine extension to a three dimensional problem. And therefore, this is very strong approach where you can routinely handle three dimensional flows. It is not that easy to extend the stream function what is a three dimensional flows to three dimensional case. Now, let us discuss a bit on boundary conditions before we close this lecture and close the module on incompressible Navier Stokes equations.

(Refer Slide Time: 09:15)



So, we will discuss briefly about how boundary conditions issues can be handled when we are talking about primitive variables. So, we have cited a problem over here that we want to model a no slip wall for a viscous flow problem. Let us say you are trying to solve a very standard benchmark problem where we have a square cavity filled with a Newtonian fluid. And then the three sides of the cavity, let us say east, west and south sides of the cavity are static while the upper wall of the cavity is moving towards the right with some velocity.

And because it is a viscous flow, what happens is the flow gets dragged by this wall as it moves and then that drives the flow through this cavity setting up some kind of vortex structure like this. So, this is often called as the lid driven cavity problem. So, as you can see there are solid walls confining the flow and therefore, modeling solid walls in viscous flow problems is always an issue.

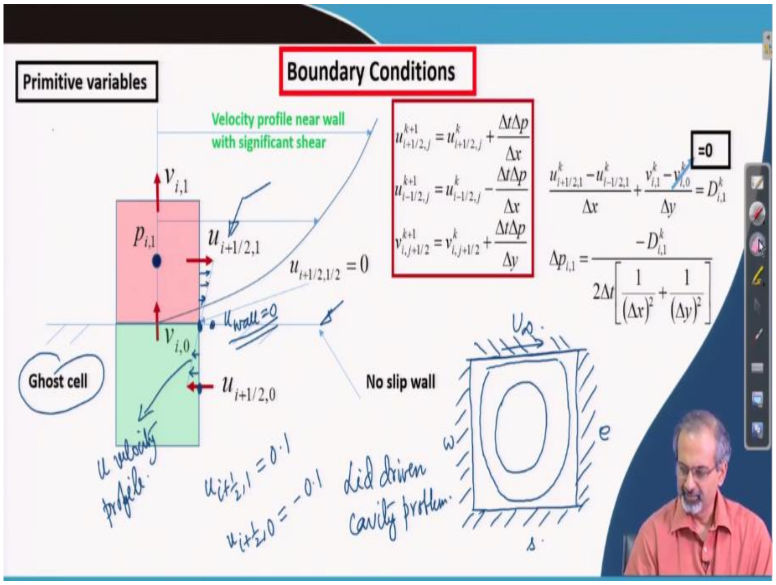
You have to handle suitable boundary condition issues so, that the solid wall is actually numerically modelled properly numerically posed properly. Otherwise, the solver would never be able to figure out that you are trying to mean that there is a solid wall lying somewhere. So, let us say we want to model a solid wall line in this location which we have drawn as a horizontal line.

Main problem is how do we make the flow solver understand that there is a solid wall line there. So, if we draw a cell sitting next to the wall, then what we mean to say is that the bottom side of the cell, say the south face of the cell is actually lying on the solid wall. Now, as you can understand in the staggered grid framework, the velocity v can be set equal to 0 on that wall, because the v is located that way.

So, normally, if this cell index is i 1 then what we have on its right is i plus half one and what we have on the top is i half (typic) one and a half rather typically. Right, so, if I go to the south face, then this can be called us, let us say half if I have called it as one and a half on top and then I can comfortably set v i half equal to 0 on the wall to explain that, that is a solid wall. But, the solid wall needs both u and v to be 0 that is what defines a no slip wall in a two dimensional problem.

So, how do I accomplish it in terms of the u? That is the problem that we have in hand. So, let us try to figure out how that can be explained.

(Refer Slide Time: 13:27)



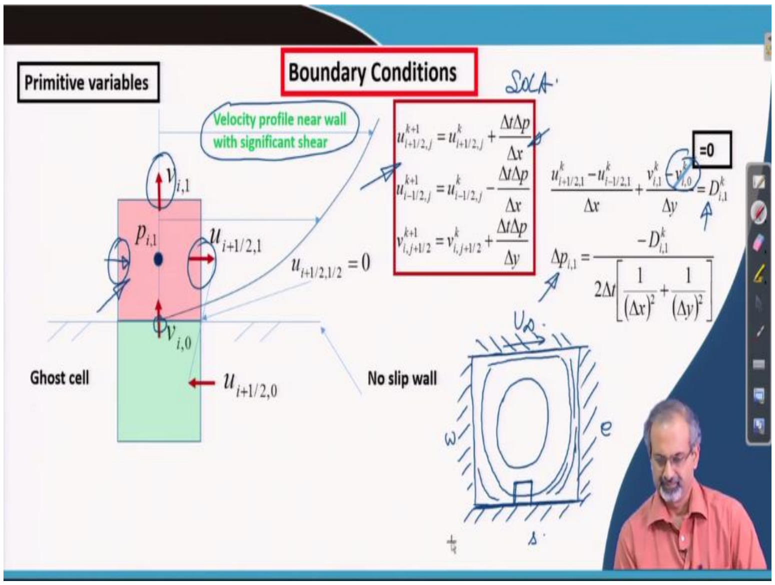
So, we have a certain value of u at say i plus half one. So, we need to have the value of u at the wall equal to 0. However, we do not have any point located on the wall where u is actually saved, because u is offset from the wall. So, in this case, a very standard technique is to define a ghost cell line below the boundary. So, though it physically does not exist, it is just a numerical existence, which can be convenient for us.

So, in that ghost cell, I said it is east side u value to a value exactly balancing the value here, but with an opposite magnitude. That means if u i plus half one has a certain value let us say 0.1 normalized or non dimensional. Then I set u i plus half zero equal to minus 0.1. What does it achieve for me? If I linearly interpolate these two values, and if these two velocities are equally distant from the wall, then right at the wall where the profile crosses over the profile attains a value of 0.

So, this profile is the u velocity profile between the cell line above the wall and the cell line below the wall. So, what have I achieved? We are just trying to show through these arrows what we have achieved. We have achieved a linear distribution of this kind which remains positive above the wall goes exactly equal to 0 at the wall and we have negative velocities below the wall in the ghost cell.

We do not really need to think much about the practical or physical existence of the ghost cell. But it is a numerical existence which helps us to enforce the u value equal to 0 at the wall. That is the beauty. So, this is a very common strategy that we deploy for modeling no slip walls when we handle primitive variables and as you can understand that if you can do it successfully, then when you simulate the flow, you will get a profile looking like this for example.

(Refer Slide Time: 16:18)



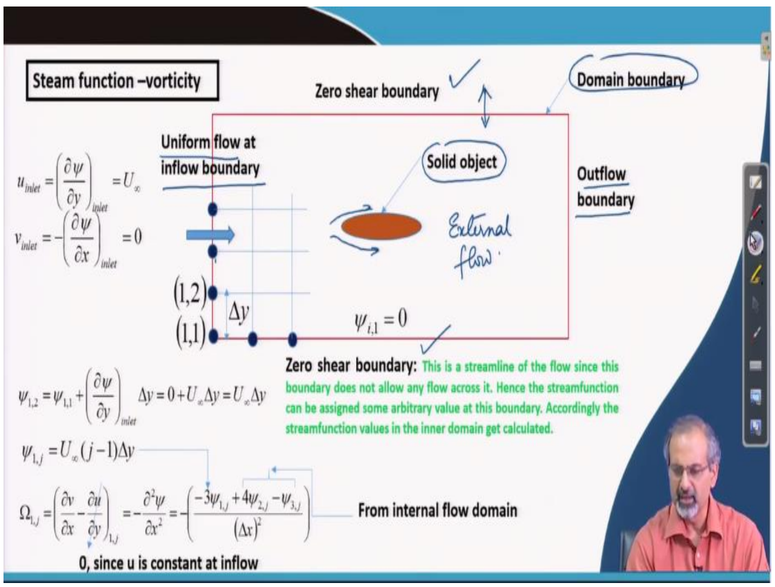
It looks like a boundary layer. That means at the wall it has gone exactly to 0. And then it is gradually rising as you go away from the wall. Because of a very heavy shearing action next to the wall, the velocity gradually increases till it reaches higher velocities away from the wall. So, as you can understand the flow moving within this lid driven cavity will be moving past walls of this kind.

And then you would have to define your cell like this somewhere here and in for the boundary condition like the way we discussed in order to make sure that this bottom wall gets numerically simulated in a proper manner. As far as the velocity and pressure corrections are concerned in this cell that we were discussing about these are the equations according to the SOLA algorithm, if you recall the k + 1th iteration level equations.

We only need three equations to be written here because one of the velocities is by default 0 on the bottom surface the south surface of the cell, the v always remains 0. So, we do not really need to iterate or update it. We just iterate or update the remaining velocities and accordingly even the divergence equation would not contain one of them on one of the velocities because that remains 0. Only the remaining velocities would contribute.

That means you have to account for how much mass comes into this volume and how much leaves only through contribution of these three velocities. So, this, this and this, they will essentially decide how the dilatation goes to 0 and accordingly the pressure gets corrected. Let us look at a bit of stream function vorticity approach also in terms of boundary conditions. So, let us say we have a rectangular domain like what we have indicated using a domain boundary here.

(Refer Slide Time: 18:10)

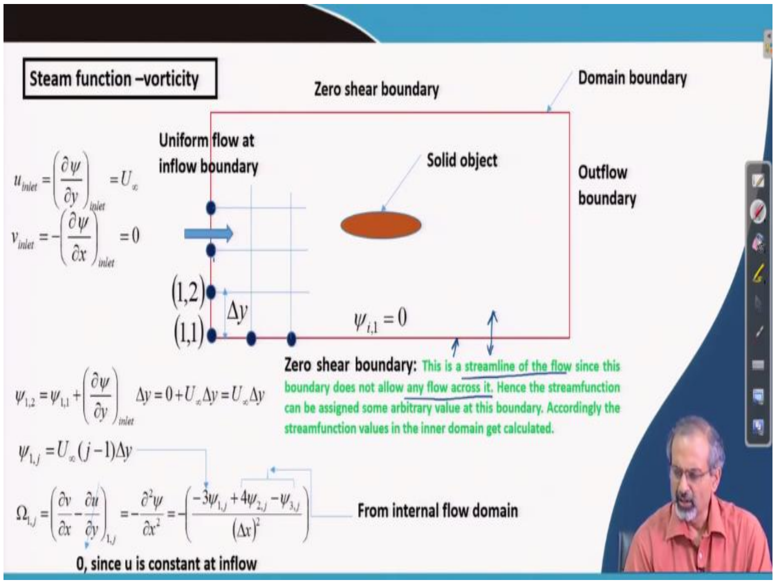


And we have some solid object inside the domain over which some flow is taking place. So, it is an external flow as we say. Because the flow moves past the body it does not enter the body and the body is apparently immersed in unconfined flow domain. So, we have an inflow boundary through which uniform flow is coming in. We have an outflow boundary which is much behind the body in the wake of the body.

And then we have lateral boundaries which we are putting as zero shear boundaries. So, that means in those boundaries, the velocity gradients become trivially small and there is no mass exchange happening normal to that boundary. This would be possible to achieve only if these boundaries at are at reasonably large distance from the object. If you keep it too close to the object, it is rather difficult to achieve these conditions at those two lateral boundaries.

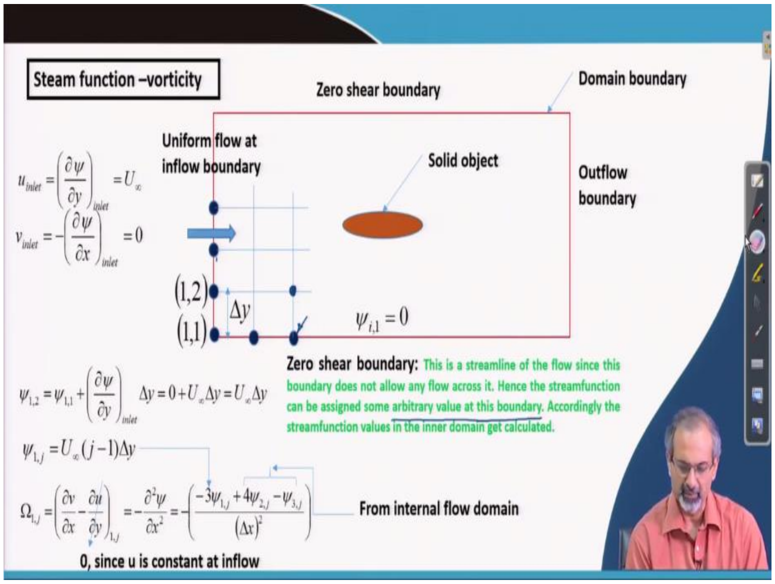
Now, if they are at reasonably good distance, then what happens is that the streamlines could remain aligned to those zero shear boundaries and that means that boundary itself becomes a streamline of the flow. Okay, you have to of course, ensure that it actually happens that way physically the dimensions should be such that we are able to go close to that approximation. Only then can we set that boundary as a stream line of the flow. Otherwise we do not try posing that condition we modify the condition there.

(Refer Slide Time: 20:12)



So, if in case we are able to use that we make this boundary as a streamline of the flow and this boundary cannot allow any flow across it only then can it become a streamline of the flow by definition. Because, if in case there is some mass exchange here, then this boundary cannot be treated as a streamline of the flow because streamlines do not allow mass flux across them.

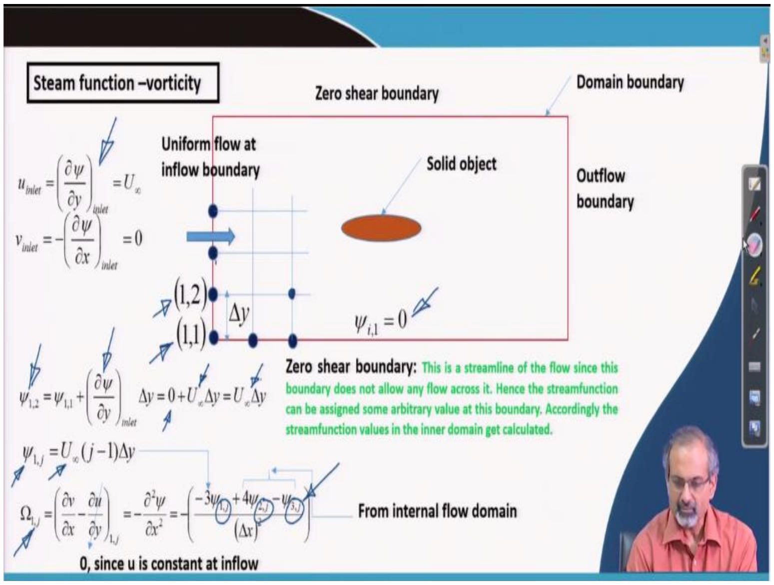
(Refer Slide Time: 20:41)



So, once it becomes streamline of the flow, we can assign some arbitrary value of the stream function at this boundary. Let us say psi i, 1 equal to 0 at each and every grid point that condition will hold good. Now, once you have done that, you can move from say this point which is lying on the stream line to the point above and then the stream function values can be updated based on existing values of the boundary condition that you have over there, you can update the values in the internal part of the domain.

And of course, remember here in the stream function vorticity approach, we make some reference to the primitive variable values also at times in enforcing boundary conditions, but, that may be only for setting boundary conditions. Otherwise, all the values have to be dealt in terms of psi and omega only. Now, in the inflow boundary, if we say we want to set uniform flow condition, then we may say that the u inlet that is del psi del y at u inlet is equal to U infinity and v inlet minus del psi del x at inlet is equal to 0.

(Refer Slide Time: 21:49)



So, if we take up the point 1 2 and 1 1, how are the stream functions going to be related there? So, psi 1 1 is 0, because that forms a part of this boundary psi i 1 equal to 0. And then, in order to go to the point 1 2, you have to add del psi del y inlet times delta y. Delta y is the grid spacing that brings in the psi change as you move to the next grid point. So, that will make it U infinity times del y. So, that is essentially the value of psi at 1 2.

If you go to a jth level, then what will happen is U infinity will get multiplied j – 1 number of times then further multiplied by delta y to give you the psi value at 1 j. That means there is a systematic increase in the stream function as you cross equal lengths and you move across the flow and every time the increment remains the same, if you are crossing one grid spacing because the velocity is constant you are crossing a uniform flow.

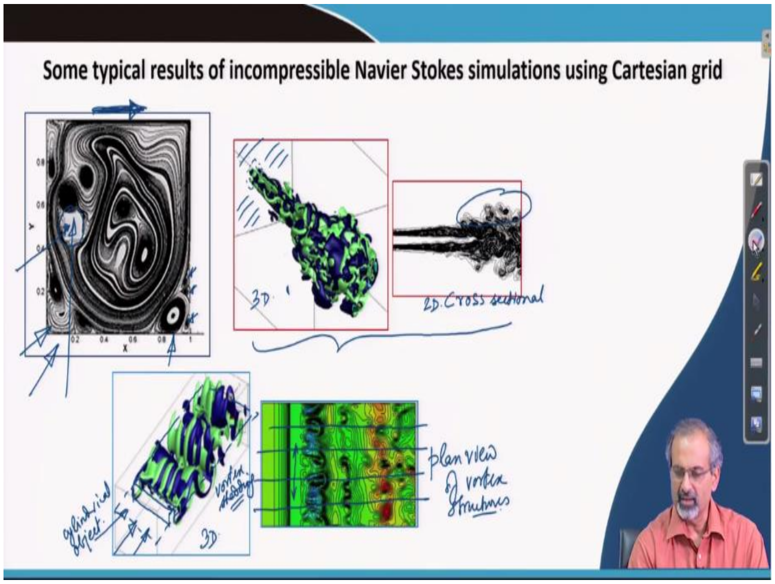
So, the psi the rate of increase of psi is constant. So, this is how you can set the inflow condition in terms of psi. Then somebody may be interested in finding out the omega let us say. At the inflow boundary, how do we do it? So, let us write an expression for omega 1 j. So, by definition it is del v del x minus del u del y 1 j. And as you can understand del u del y will not change because u is constant in this boundary it is U infinity.

So, only thing that will contribute in this case is del v del x. And del v del x is minus del 2psi del x square. So, what do we do? We invoke the values of psi at the boundary and values which lie inside in order to compute the del 2psi del x square. So, it is a one sided difference. So, we have done all of this earlier in finding you know, centrally different schemes or one sided different schemes.

So, here we have essentially employed an approximation for the second derivative with a one sided scheme. So, that we make use of psi values at the boundary, the next grid next to the boundary and the second level next to the boundary. So, 1 j 2 j and 3 j with suitable coefficients give us the value of the secondary derivative of psi with respect to x. And that is nothing but the omega at the boundary.

So, this is how we compute the psi's and omega's at the boundaries when it comes to the stream function vorticity approach. So, we cited only a few examples, however, the student can work out on many others by looking at different kinds of problems in fluid flows. Before we end, we will just show glimpses of some results using two dimensional or three dimensional incompressible Navier Stokes simulations on a Cartesian grid, like the ones we have discussed in this module.

(Refer Slide Time: 25:35)



So, we were earlier discussing about the lid driven cavity problem. So, you can see streamlines of a similar problem, but with an obstacle placed inside the cavity. So, this is an obstacle inside the cavity. But, the cavity still moves with the top wall sliding from left to right, while the other surfaces of the cavity are stagnant or static. And you can see very rich vortex structures developing inside the cavity.

And additionally, because there is an obstacle, the obstacle is also responsible for generating some more vortex structures. So, you may be aware of von Karman vortex shedding and things like that. So, you can actually see a bit of shedding occurring in the vicinity of this obstacles wake. Additionally, some water structures are developed at corners and if you have high Reynolds number flows moving through these cavities, then you can see very rich structures.

So, there are primary vortices there are secondary vortices there could be tertiary vortices. So, if you go to higher Reynolds number simulations usually in Navier Stokes simulations, you will see that larger structures are accompanied by smaller structures are accompanied further by still smaller structures and so on. So, there is as if a cascading of vortex structures. So, this these are very interesting phenomena which can be explored if you have a simple two dimensional incompressible Navier Stokes solver.

So, we have discussed enough about the tools which are required to develop such a solver. So, I would certainly encourage you to look at possibilities of trying to develop a code of your own so, that you can do this or attempt these simulations on your own. Of course, you have to also work on boundary conditions. That is why we touched on the necessity of boundary conditions in the last few slides of this lecture.

And here you see some typical three dimensional flows. So, this is an open jet. So, there is some kind of an orifice here located on a wall a solid wall from where some mass is emanating into neighboring say air, which is stagnant. And then you can see very rich vortex structures even in the regions where the jet interacts with the neighborhood. And there are some instabilities which generate which lead to formation of such vortex structures.

We have the Kelvin Helmholtz instabilities which generate at shear lengths because the high velocity jet is shearing the flow where it is interfaced with the stagnant flow that is very heavy shear and that leads to instabilities in the interface. So, such instabilities can trigger breaking up of that shear layer into once which have very rich vortex structures. So, this is a cross sectional view a 2d cross sectional view while this is a three dimensional view of the check.

In this problem, you have a cylindrical body of this kind spanned across the domain again this is the these are three dimensional flows. So, this is typically a cylindrical object which is spanned across the flow. So, the flow is coming in this direction impinging on the object and after that there is Karman vortex shedding in the wake of the object. And if you look, if you take a cross sectional view by putting a plane like this in the wake then this will give you a kind of a plan view of vortex structures.

And then you can find that the vortex structures, they have a spanwise distribution, they are not identical in the spanwise direction. The wake is three dimensional, as we say. So if the wake was perfectly two dimensional, then each plane like this would have given you the same kind of structures that does not happen. So, there is more to all this, but it is far beyond the scope of the introduction to CFD.

So we will stop here. But I will once again encourage the participants to look closely into the techniques that we have discussed and start developing the habit of working on solvers on your own instead of just opting for commercial solvers and things like that. Thank you.