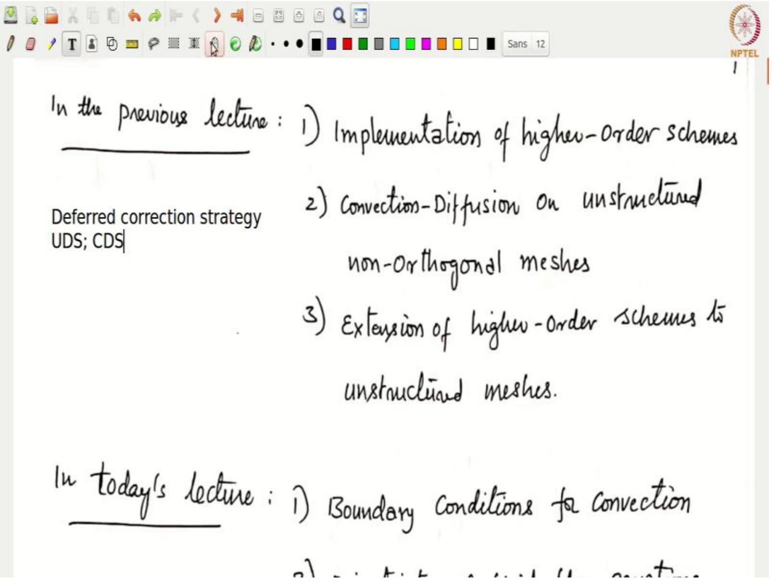
Computational Fluid Dynamics Using Finite Volume Method

Prof. Kameswararao Anupindi
Department of Mechanical Engineering
Indian Institute of Technology, Madras

Lecture - 34
Finite Volume Method for Convection- diffusion and fluid flow calculations

(Refer Slide Time: 00:14)



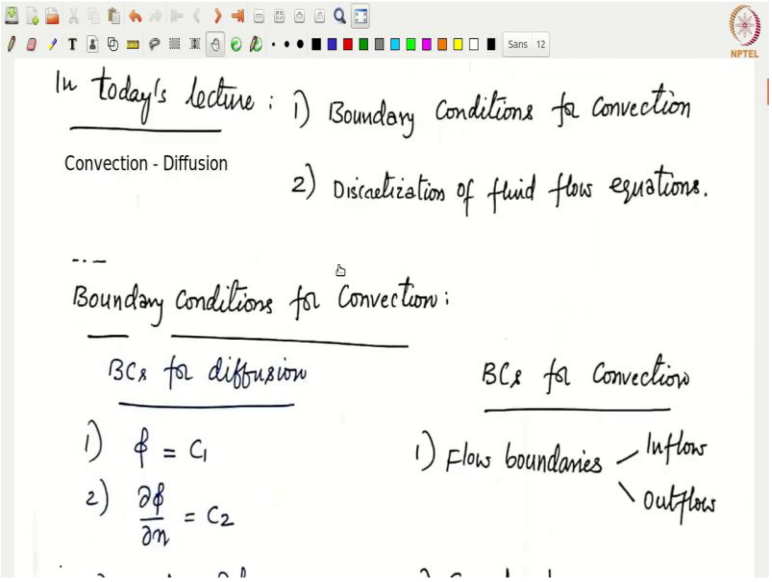
Hello everyone, welcome to another lecture as part of our ME6151 course, Computational Heat and Fluid Flow. So, let us get started. So, in the previous lecture, we looked at the implementation of higher order schemes right whether on a structured mesh or on a unstructured mesh essentially using the deferred correction strategy right.

And then we looked at convection diffusion on unstructured non-orthogonal meshes, essentially how do we go about solving convection diffusion, we formulated the equations for both the upwind difference scheme as well as the central difference scheme right.

We have also extended our kind of discuss the higher order schemes right, we extended them to how to solve them on unstructured meshes. For example, both the second order accurate scheme, second order accurate upwind scheme as well as the quick scheme right. Both of them we saw how do we extend them to unstructured non-orthogonal meshes alright, excuse me, alright.

So, let us move on. In today's lecture we going complete the discussion on convection, because the only thing that is remaining is the boundary conditions for convection. So, we are going look at that. And then that essentially finishes the chapter on convection diffusion ok.

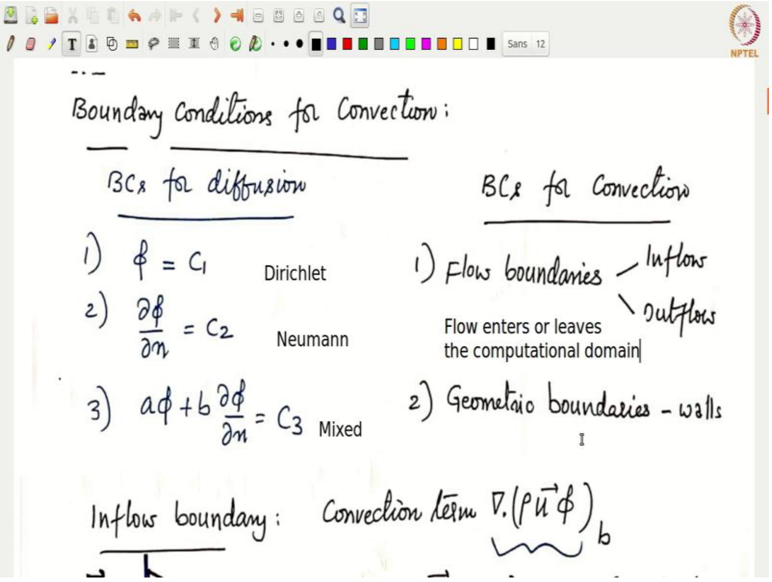
(Refer Slide Time: 01:41)



Then the next chapter is actually on the linear solvers, but most of the linear solvers part we have already discussed. So, we will skip that for now and we will move on to the chapter afterwards that is the final chapter that is basically the discretization or the solution of fluid flow equation, so that is the final chapter.

So, once we finish the fluid flow equations, we will come back to the linear solvers chapter and finish whatever is remaining there alright ok. So, let us move on with the boundary conditions for convection.

(Refer Slide Time: 02:15)



So, let us recall the boundary conditions we have for diffusion were around three types of boundary conditions right. We said if you have only a diffusion problem, whether it is steady or unsteady, you would have you specify the value of the unknown that we call it as Dirichlet boundary condition. If you specify the gradient of the unknown, then we said this is the Neumann boundary condition.

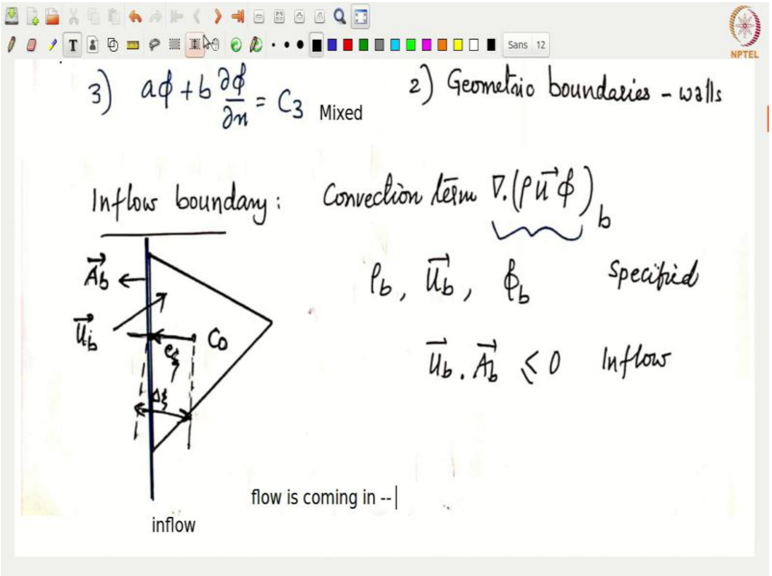
And the third type of boundary condition, we can have is basically a combination of Dirichlet and Neumann that is basically a phi plus b partial phi partial n equal some C 3 or something that is basically your mixed boundary condition ok. So, this is these are the boundary conditions for if you have only a diffusion problem. Now, if we have a convection problem or a convection diffusion problem, then you need to specify few boundary conditions and those kind of fall into two categories.

One of them belongs to the category of flow boundary conditions on the flow boundaries, that means, these are the boundaries where the flow can enter the domain or leave the domain ok. So, this is basically the boundaries where flow enters or leaves the computational domain ok, computational domain. And we kind of categorize them as inflow boundaries and outflow boundaries. So, we need to see how do we discretize the cells which are sharing a face on the inflow boundaries or on the outflow boundaries.

And we have another type of boundaries that exist which are known as geometric boundary boundaries ok. So, these are basically walls. For example, the walls of a duct in which the flow is fluid is flowing ok. So, these are essentially the geometric boundaries.

So, we need to see how do we discretize the convection term on those faces which happen to be on the geometric boundaries ok, essentially these are the cells which have a face on one of these boundaries ok. So, we will see how do we apply the boundary conditions for these cells here alright. Let us move on then.

(Refer Slide Time: 04:18)



So, let us first start off with inflow boundaries. So, inflow boundaries, so let us essentially we have a schematic that is drawn here. This is basically an unstructured mesh. We take this cell and this vertical line here is an inflow boundary ok. So, we say this is an inflow boundary. So, out of the three faces, two faces are interior faces and this one is a boundary face. Just like before we will call it as C 0 cell, and the vector connecting the cell centroid C_0 to the face centroid is e_ξ.

And the face of this boundary face is basically the area vector is pointing outwards like any other face. And then we also have u b bar which is the inflow velocity vector that is pointed towards the into the cell ok. Now, if you look at the convection term for this particular face, this particular face will have both the convection and the diffusion terms. Now, we are looking at the convection term. The convection term would read as del naught rho u bar phi b right.

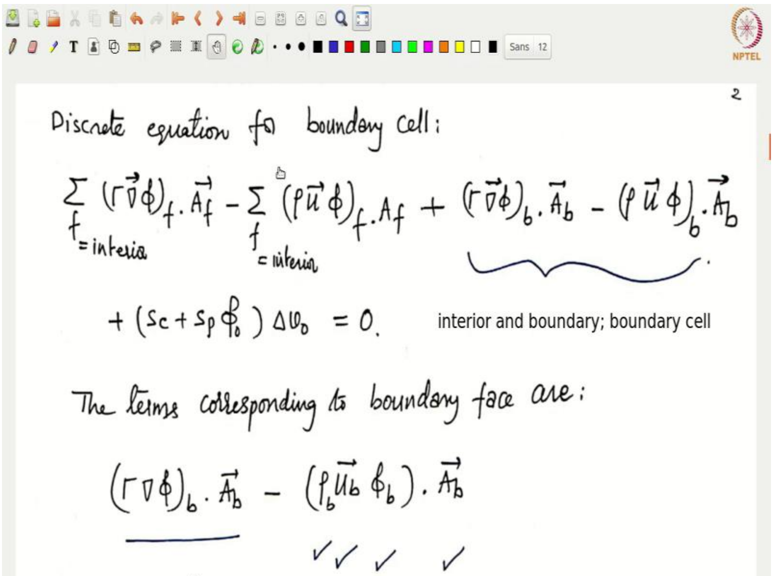
Now, what needs to be specified on this boundary on the inflow boundary? What needs to be specified essentially, because the flow is coming in we need to specify essentially everything that we need here right because the flow is coming in.

So, we need to know what is the density of the fluid that is coming in, with what velocity the fluid is coming in, and what is the scalar that it is bringing in right; so, all these three needs to be known. For example, if you are solving for a convection diffusion equation, then u bar by default is anyway known right. The flow field is already known.

So, we need to know of course, you know what is the scalar value that it is brining is brought in through the inflow boundary. So, essentially we need to specify. So, somebody says if there is an inflow boundary, then the density on the boundary, the velocity vector on the boundary as well as the ϕ_b the scalar value on the boundary, all three needs to be specified ok. So, once we know all these three, of course, we can evaluate this term right.

So, then how do we call a boundary as inflow boundary? A boundary will be inflow boundary if you are will you the velocity vector is kind of pointed inwards into the domain. So, that means, $\vec{u}_b \cdot \vec{A}_b$ has to be less than or equal to 0, then only we can call this as an inflow boundary right. So, far so good, then let us move on to how do we kind of put these terms back into the equation.

(Refer Slide Time: 06:42)



So, the discrete equation for boundary cell would be basically you have the diffusion terms right. We have the integral terms which are the $(\Gamma\nabla\phi)_f \cdot \vec{A}_f$. And we have the convection term that is $(\rho\vec{u}\phi)_f \cdot \overrightarrow{A_f}$. So, these are for the interior faces, that means, in the triangular cell here these are basically the two interior faces ok. And then we have not written the boundary face here.

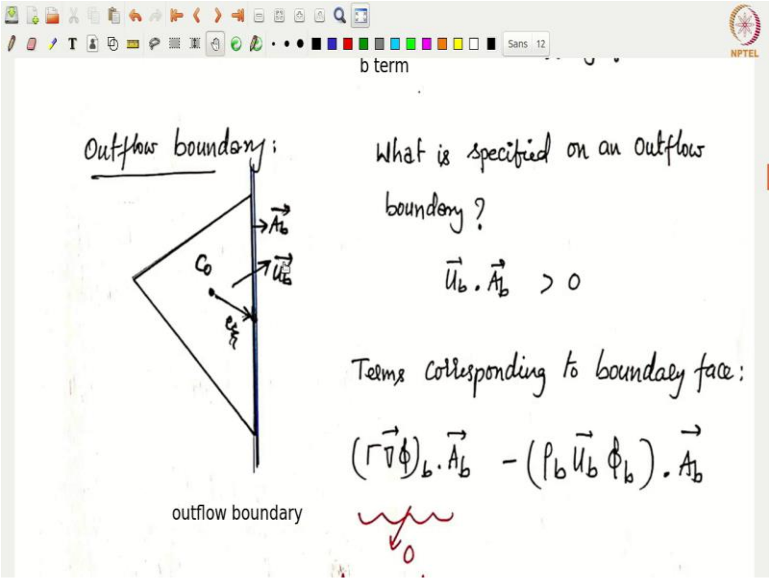
So, if I write the boundary face here, you have $(\Gamma\nabla\phi)_b \cdot \overrightarrow{A_b}$ minus $(\rho\vec{u}\phi)_b \cdot \overrightarrow{A_b}$, so this is the only for the boundary cells plus we have the source term that is $S_C + S_P\phi_0$ times ΔV_0 equal to 0 ok. So, this is our original regular equation, but written in terms a kind of separate it out in between the interior and the boundary and boundary. So, this is basically written for a boundary cell alright.

Then what about the, so we know how to kind of discretize these interior faces already. Now, we just need to look at how do we discretize the boundary terms right. Only thing that needs to be understood is how do we discretize these two terms right alright. That means, if we look at those two terms, even the first part which corresponds to the diffusion is already known to us. We know how to discretize this.

This will basically give rise to some coefficient times $\frac{\partial\phi}{\partial\xi}$, and some other coefficient times $\frac{\partial\phi}{\partial\eta}$ if you have an unstructured non-orthogonal mesh. Now what about the other term?

So, $(\rho\vec{u}\phi)_b \cdot \overrightarrow{A_b}$ in which rho b bar is known sorry ρ_b is known, the density at the boundary is known, velocity is known, phi is known, and $\overrightarrow{A_b}$ is known right. So, we know essentially everything, everything is known to us. So, we can just plug it in this value, and then calculate evaluate the particular value of this particular term ok.

(Refer Slide Time: 08:44)



Now, where will this term go? Because everything is known, this term will has to go to the b term of the equations alright. In fact, for that matter, even this will going to the b term is not it? So everything will going to the b term. So, that is how we go about implementing inflow boundary conditions for convection diffusion ok. Let us now move on to the other boundary condition which is basically an out flow boundary.

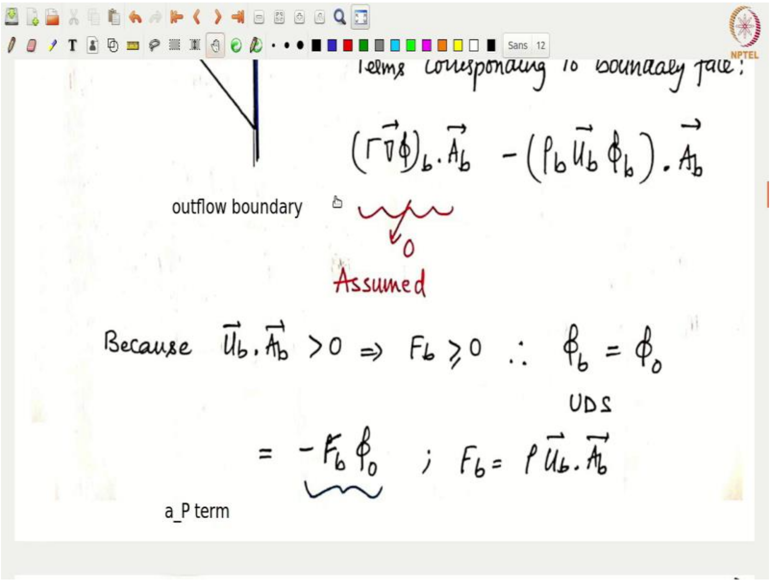
Let say if we have an outflow boundary, that means, we call this particular cell as again a cell that has a shades a face on the outflow. So, this vertical line here is the outflow boundary. As before we have a C_0 cell, we have e_ξ, the velocity vector of the boundary face is, so the area vector is pointing away, and the velocity vector is also is drawn here such that the flow is leaving from the cell out of the domain ok.

So, on the left hand side, here is where we have the domain and this is the boundary alright. Now, what do we, when do we call this as an outflow boundary? We can call this as an outflow boundary if your $\vec{u_b} \cdot \vec{A_b}$ is what, is positive right? So, both of them have to be in the same direction if that is positive, then we can call this as an outflow boundary.

If these two, if the dot product of $\vec{u_b}$ and $\vec{A_b}$ is negative, then it is an inflow boundary ok. Now, what needs to be specified in outflow boundary? What do we need to specify? Do we know anything? We do not know we cannot specify anything because whatever is happening inside has to leave the domain.

So, the outflow boundary is kind of a boundary where things happening inside, needs to be kind of dictate what is what should actually leave through the outflow right depending on the flow rates and things like that ok. So, essentially we do not know yet what needs to be specified essentially, we cannot specify any thing as of now, because it we feel that it whatever is kind of computed inside needs to kind of dictate what should leave through the boundary ok.

(Refer Slide Time: 10:45)



Now, what about the terms that correspond to the boundary face? We can write a similar equation for this particular cell where we have contribution of the interior faces and contribution of the boundary face.

We will only look at the boundary face. The boundary face is basically you have $(\Gamma\nabla\phi)_b \cdot \overrightarrow{A_b}$ minus $(\rho\vec{u}\phi)_b \cdot \overrightarrow{A_b}$ right. So, you have the diffusion part and the convection part. Now, for most of the fluid flow problems that we encounter, we assume that the diffusion component is 0 ok. So, we make this assumption. We will come back to what is the consequence of this assumption ok.

So, we assume that there is no diffusion happening in the cell that is sharing a face on the outflow ok. So, we are assuming that this is 0. So, the only term remaining is the convection term as for as this is concerned. By the way the diffusion components for corresponding to these two interior faces are still intact ok, they are not assumed to be 0; only the component related to the boundary face is assumed to be 0 ok.

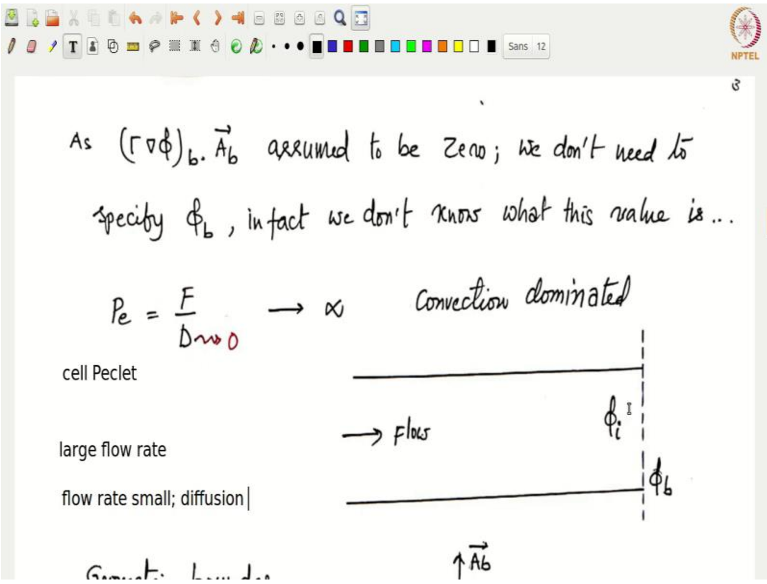
Then we only have the convection term that is remaining, but because we know what is $\vec{u_b} \cdot \vec{A_b}$ right $\vec{u_b}$ is known from the flow field right because this is a convection diffusion equation.

So, $\vec{u_b}$ is already known. $\vec{u_b} \cdot \vec{A_b}$ is positive we know that means, the flow rate that we have is positive. Because this is positive what will be the value for ϕ_b? Because this is positive if I use an upwind difference scheme, the ϕ_b has to be equal to ϕ_0 right if we use an upwind difference scheme ok.

Then we can rewrite this equation as minus $(\rho\vec{u}\phi)_b \cdot \vec{A_b}$ as minus F_b times ϕ_0 right, because F_b itself is $(\rho\vec{u}\phi)_b \cdot \vec{A_b}$. So, this is basically your now convection term which is essentially evaluated for an outflow boundary ok. Now, where will this term go? This term has to go into the, the will it going to the b term?

No, it will going to the a_P term right essentially because you have ϕ_0. So, essentially this has to going to the a_P term or for the C_0, the central coefficient right alright. Now, let us kind of discuss further on this particular assumption that we have made that the diffusion is neglected for the boundary face when compared to convection ok.

(Refer Slide Time: 13:15)



So, let us look at that. So, because we have assumed this to be 0, we do not need to specify the value of ϕ_b right on the boundary. In fact, we do not even know what this value of ϕ_b will be, because it has to be dictated from what is come what is going on inside the boundary right.

By making this assumption that the diffusion is 0, what we made is we go back to our cell Peclet number discussion, so we said that the cell Peclet number that we have kind of goes to infinity, because the diffusion the ratio of convection to the diffusion.

And the diffusion we have assumed to be 0 for the local cell Peclet number for the particular face, then the local cell Peclet number is now tends to infinity right, that means this is a convection dominated flow right where we are assuming that the affects of diffusion are negligible only for the boundary cell between the boundary cell and the boundary face alright.

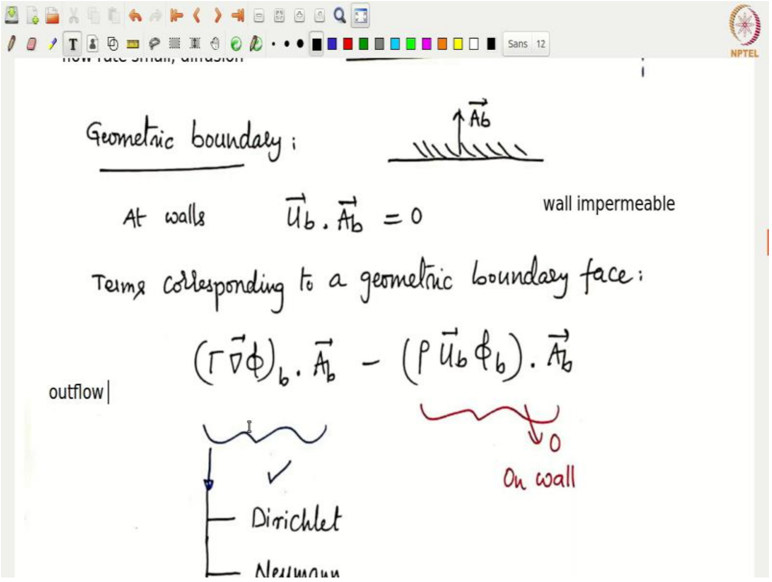
Now, what the what does that mean? That means that let us say if you have a flow and this flow rate is let us say is quite large we have a large flow rate, that means, the convection is very very much dominant. As a result this phi i that we have which is the value of phi in the interior will get convected to the boundary right. So, ϕ_b as a result will take a value of ϕ_i; and ϕ_i would not be affected because of ϕ_b right.

Because of the large flow rate whatever is happening inside would be dictated by what will happen at the boundary right, but not the other way around. So, for all these cases where the flow rates are kinds of considerably large or significant compared to diffusion, then we can assume that the diffusion component is very small.

However, if you realize that the flow rate is very small and if the diffusion plays a role, then we of course, need to specify what is the value of ϕ_b. Because the value of phi b would diffuse and affect the value of the cell centroid near the near boundaries for near boundary cells ok, so that needs to be accounted for.

But otherwise for most of the problems for the convection operated problems, whatever is happening inside can be taken to be same as what will happen will be convected downstream and the diffusion affects between the cell centroid to the face are quite small alright. So, that is about the implementation of outflow boundary conditions.

(Refer Slide Time: 15:38)

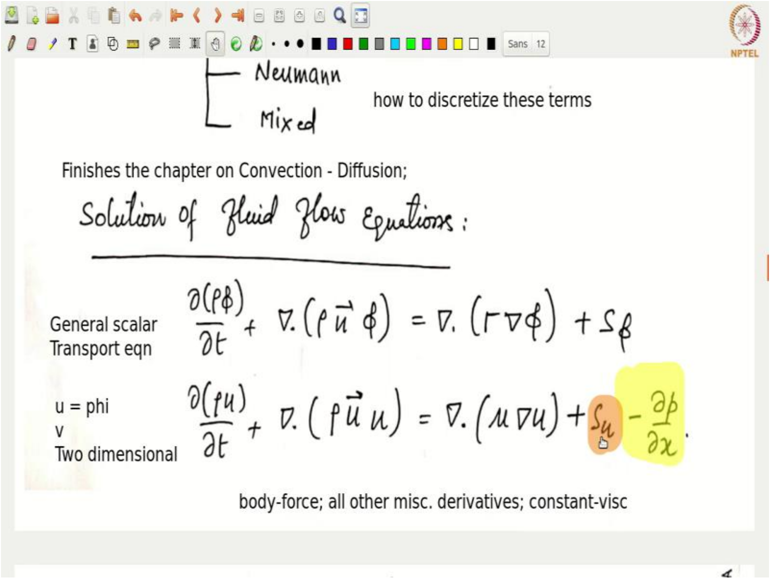


Now, we have the final one more boundary condition that is basically your geometric boundaries, that means, we have walls such as this in which we have a flow that is going in. And the let us say there is a cell here we have a cell here, and this is your $\overrightarrow{A_b}$ again pointing away from the away from the cell ok. Now, at walls essentially the condition we have is the flow the flow cannot go normal to the face right. It cannot going normal to the face, that means, the flow is kind, so the wall is kind of impermeable right.

So, for the flow, that means, the dot product of the velocity and the area vector that is $\overrightarrow{u_b} \cdot \overrightarrow{A_b}$ would be equal to 0 right. So, here we have not made any assumption related to the viscosity. We have not made anything related to the related to the no slip condition ok. We have not made any assumption, but we are only talking about the impermeability of the wall which makes it $\overrightarrow{u_b} \cdot \overrightarrow{A_b}$ equal to 0 ok.

So, once you have $\overrightarrow{u_b} \cdot \overrightarrow{A_b}$ equal to 0, what will be the flow rate across the wall that is of course 0. That means, if we look at the boundary contribution, then we have the diffusion term and the convection term, because the flow rate is 0, this term will go to 0 right. And we are only now left with the diffusion component.

(Refer Slide Time: 17:03)



So, this is kind of opposite behaviour to what we saw in the context of outflow boundaries right. Because for outflow boundaries, we said we are neglecting diffusion and the only thing we have is the convection; whereas for geometric boundaries we have convection equal to 0 and we only thing we have is the diffusion So, these are two kind of opposite elements.

Then what about the diffusion component, the diffusion terms? We know that again somebody has to specify that it is as a Dirichlet boundary condition or using a Neumann boundary condition or with using a mixed boundary condition, and these can be now evaluated depending on the boundary condition that we already know how to essentially discretize these values right in order to discretize these terms whether it is a Dirichlet Neumann or a mixed boundary condition ok.

So, that kind of finishes the chapter on that kind of finishes the chapter on convection diffusion so alright. So, let us kind of move on to the solution of fluid flow equations. So, in fact, if you look at the general scalar transport equation that is your general scalar transport equation ok, basically that is given by $\frac{\partial}{\partial t}(\rho\phi) + \nabla \cdot (\rho\vec{u}\phi) = \nabla \cdot (\Gamma\nabla\phi) + S_{\phi}$ right.

We have now seen each and every component of this equation. We have seen how to solve the diffusion part, the convection part and the unsteady part. So, in principle, we can now solve for the convection diffusion equation or a general scalar transport equation for any scalar phi ok.

Now, if you look at the fluid flow equations, we basically have pretty much a similar structure as compared to the general scalar transport equation. Only thing is that now your ϕ is replaced with u right and v. So, instead of one equation, now we have two equations.

Let us say if we consider a two-dimensional flow situation ok. If we consider two dimensional flow situation, then we have instead of one equation we have two equations. One for the x-momentum equation; the other one for the y momentum equation, in which ϕ is replaced with u or v ok.

Of course, we also have continuity equation in addition to these two equations ok. So, if we compare what we have is the first term is $\frac{\partial}{\partial t}(\rho u)$ plus $\nabla \cdot (\rho\vec{u}u)$ right. Excuse me. So, we have ϕ replaced with u, and then $\nabla \cdot (\mu\nabla u)$ gamma replaced with μ, instead of $\nabla\phi$ we have ∇u plus S_u minus $\frac{\partial P}{\partial x}$. Now, this is the something this is the term which is extra here right as compare to what we have in the general scalar transport equation ok.

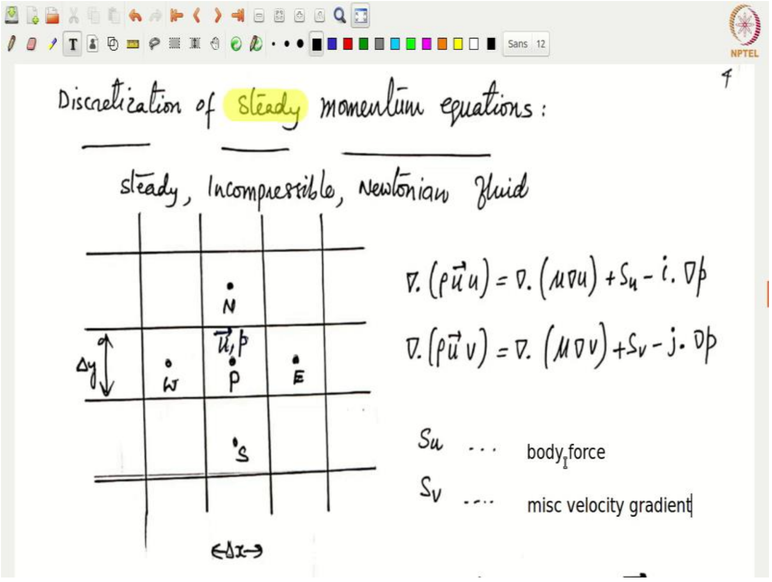
So, this prescribed term is something that is extra which we need to see how do we do it, how do we evaluate this. But otherwise this looks very similar to the general scalar transport equation. Of course, we also realize that the term that we have here this is basically your S_u right this is we have written it as source.

But this contains what? This contains the, this contains the not only the body force term right, but also it contains all other the miscellaneous derivatives that were coming out which were kind of not written here right. It contains several velocity gradient terms as well.

Now, we know that if we assume a constant viscosity right, if we assume a constant viscosity and if we assume the flow to be incompressible, then we know that S_u will have only the body force component, because every other term the miscellaneous derivatives of velocity gradients would all go to 0 right. They will all go to 0 by because of the conservation of mass equation ok.

So, we realize that alright, but any way those will not cause an issue in terms of the solution algorithm whether we have constant viscosity or not, it does not matter. We can still go ahead with the calculation of the source term by incorporating both the body force, and all other miscellaneous velocity gradient terms alright.

(Refer Slide Time: 21:30)



So, how do we now discretize the momentum equation? So, we would like to get kind of get started with a steady momentum equation ok. We do not want to do the unsteady as of now. So, let us make some approximations. The approximations we make are basically the flow is steady, and it is also incompressible, and the fluid we assume is basically a Newtonian fluid that means whose shear stresses are proportional to the strain rates ok.

So, we make these assumptions, and then go ahead with the solution of the fluid flow equations. We also assume to begin with a uniform or a structured mesh which is basically given by our regular Cartesian mesh basically that is cell P, and we have east and west cells, and the north and the south cells ok. And the width of the cell P for which we want write the discrete equations is basically given by Δx times Δy ok.

Then what are the equations we have? If we consider steady flow, we do not have the $\frac{\partial}{\partial t}(\rho u)$ term is gone. So, only thing is we have the convection term that is $\nabla \cdot (\rho \vec{u} u)$ equals the diffusion term is $\nabla \cdot (\mu \nabla u)$ plus S_u minus we have essentially minus $\frac{\partial P}{\partial x}$ right, minus $\frac{\partial P}{\partial x}$ that I have written it as in terms of ∇P. We can write it as $-\hat{\imath} \cdot \nabla P$ right. This is basically evaluates to $\frac{\partial P}{\partial x}$.

Similarly the y-momentum equation is basically $\nabla \cdot (\rho \vec{u} v)$ equals $\nabla \cdot (\mu \nabla v)$ plus S_v minus $\hat{\jmath} \cdot \nabla P$ ok, so that is your y-momentum equation which is basically minus $\frac{\partial P}{\partial y}$ ok. Now, we realize that the S_u and S_v do not just contain the body force terms. They also contain all the miscellaneous velocity gradient terms which we did not write in the viscous terms here ok.

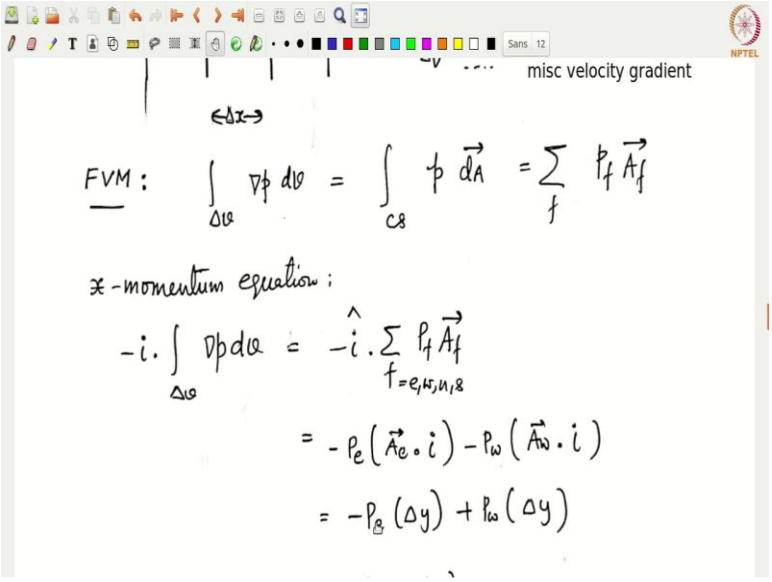
And depending on the value of the viscosity we take all other terms would actually go to 0 as well that is the that possibility is also there alright. Let us move on then. Then if we want to apply finite volume method to these two equations, we know that we can solve for these just like the way we have done before.

So, we know how to apply the finite volume method for the convection term and for the diffusion term and for the source terms. Only thing we do not know or we have not done is basically the pressure gradient term. So, let us look at only this term. Apply finite volume method for this, and then plug it back into this equation and write the complete general discrete equation afterwards ok.

So, we are only focusing right now on the grad p term. So, if we were to apply finite volume method, the first step is to integrate it on a controlled volume. So, if we take ΔV is the volume of the cell P, so this will be integral ΔV, ∇P dV right. Again invoking Gauss divergence theorem, we can write this as integral over the control surface P$\vec{u}$ that equals assuming that the pressure on the faces is constant.

And, the cell centroid value or the face centroid value can be used to represent that pressure on the face, then if this is P_e this is P_w. Similarly, P_n and P_s, then we can write this as a summation of all the faces east, west, north, south $P_f\vec{A_f}$ that is the pressure on the each of the faces times the $\vec{A_f}$ ok.

(Refer Slide Time: 25:01)

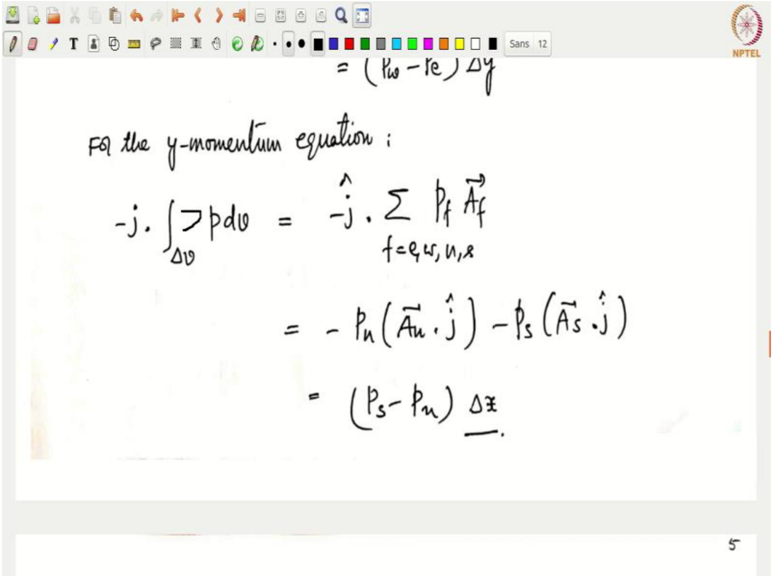


Now, if we go back to the x-momentum equation, then what we have is we have not just ∇P, we have $-\hat{\imath} \cdot \nabla P$. So, that means, whatever we got here, we need to take a dot product with $-\hat{\imath}$ ok. So, that means, going back to the x-momentum equation the term we have is $-\hat{\imath} \cdot \nabla P \, dV$ integral ΔV this is basically $-\hat{\imath}$ dot summation f, $P_f \overrightarrow{A_f}$ ok, where f is east, west, north, south ok.

Now, what about $\overrightarrow{A_f}$? What are the values for area vectors for faces? We know that $\overrightarrow{A_e}$ equals $\Delta y \hat{\imath}$; $\overrightarrow{A_w}$ would be $-\Delta y \hat{\imath}$; $\overrightarrow{A_n}$ would be $\Delta x \hat{\jmath}$, and $\overrightarrow{A_s}$ would be $-\Delta x \hat{\jmath}$ ok. So, essentially if we take a dot product with $-\hat{\imath}$ only terms that survive out of this equation are the east and west, because the north and south contain j quantities which will give you 0 when you take a dot product with $-\hat{\imath}$ right.

That means only thing survives is basically the east and west terms. So, we have minus $P_e \overrightarrow{A_e}$ dotted with $\hat{\imath}$ minus $P_w \overrightarrow{A_w}$ dotted with $\hat{\imath}$. $\overrightarrow{A_e}$ dotted with $\hat{\imath}$ is basically Δy; $\overrightarrow{A_w}$ dotted with $\hat{\imath}$ is your $-\Delta y$, basically makes this as a positive. So, what we have is Δy times P_w minus P_e ok. So, that is how you get for the pressure gradient term in the x-momentum equation.

(Refer Slide Time: 26:27)

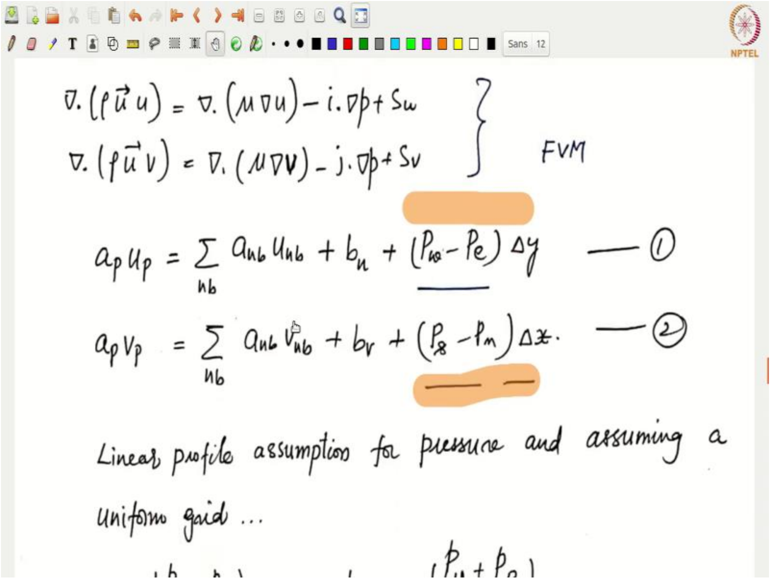


It kind of the discrete value reads as P_w minus P_e times Δy ok, that means, the value of the pressure on the faces P_w minus P_e times Δy is the evaluation of this particular quantity the pressure gradient coming up in the x-momentum equation alright. Then let us look at the y-momentum equation. So, the y-momentum equation has minus j dot this is not correct this should be a gradient this is ∇P right. So, this should be a grad here ok.

So, this is basically ∇P dV, that means, which if you apply Gauss divergence theorem, this will come to $-\hat{\mathrm{j}}$ dotted with sigma $P_f\overrightarrow{\mathrm{A_f}}$ ok. And again all the faces are east, west and north and south ok. Again we realize that because we are taking a inner product with ĵ only terms that survive here are the north and south, because they are the once which have ĵ with them; that means we get minus $P_n\overrightarrow{\mathrm{A_n}}$ dot ĵ minus $P_n\overrightarrow{\mathrm{A_n}}$ dot ĵ ok.

So, $\overrightarrow{\mathrm{A_s}}$ is basically your $-\Delta x\hat{\mathrm{j}}$, and this is $\Delta x\hat{\mathrm{j}}$. As a result this will be only s minus P_n times Δx ok. So, where pressure on the south face minus pressure on the north face times delta x is what we get alright, that kind of completes the discretization of the x and y-momentum equations, the pressure terms that we represent gradient terms that we get.

(Refer Slide Time: 28:10)



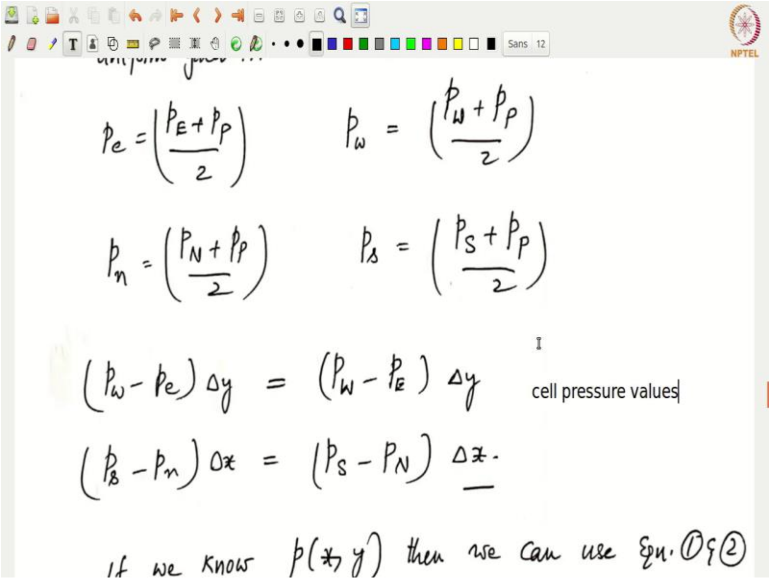
And we are now in a position to kind of write down the discrete equations. So, if we start off the steady Newtonian incompressible flow equations are given here which is $\nabla \cdot (\rho\vec{u}\mathrm{u}) = \nabla \cdot (\mu\nabla \mathrm{u}) - \hat{\imath} \cdot \nabla \mathrm{P} + S_{\mathrm{u}}$. And the other one is $\nabla \cdot (\rho\vec{u}\mathrm{v}) = \nabla \cdot (\mu\nabla \mathrm{v}) - \hat{\jmath} \cdot \nabla \mathrm{P} + S_{\mathrm{v}}$ right.

So, if we plan to apply finite volume method for this, then just like the way we have done it for the general scalar transport equation, we can write the final discrete equation as $a_P\phi_P = \sum \mathrm{a_{nb}}\phi_{nb} + b$ right. Similarly, because the quantity here we are solving for is u, so we can write this as $a_P\mathrm{u}_P = \sum \mathrm{a_{nb}}\mathrm{u}_{nb} + b_u$ ok.

So, we have this quantity. And then plus the pressure gradient term here would reflect to something like this which is basically P_w minus P_e times Δy. So, we got basically this is now our extra term right. So, basically this is our extra term as compared to the general scalar transport equation ok.

Similarly, the y-momentum equation can be written in the discrete counterpart as $a_P\mathrm{v}_P = \sum \mathrm{a_{nb}}\mathrm{v}_{nb} + v_u$ coming from the source term plus we have for the pressure gradient we get P_s minus P_n time Δx ok. Now, remember that these are basically evaluated on the faces. So, these are the face values alright.

(Refer Slide Time: 30:02)



Now, of course, can we solve for these equations 1 and 2? We cannot solve for them as it is, because these are 2 equations and the unknowns are 3 right, because your u v and pressure – 3, there are three unknowns, but these are only two equations. What is the third equation? The third equation is what we have not written down here is the continuity equation ok.

So, essentially in principle we have three equations and three unknowns. So, we can possibly solve for this thing. But if somebody gives you pressure field, if the pressure field is known then these two equations can be solved right. If the pressure field is known, then it is a matter of calculating putting it back here, and calculate from the first and second equation the u and v velocity fields ok.

So, now, remember that up till now when we had the general scalar transport equation our u was known right it was a known field, but whereas here in the fluid flow equations the u bar the velocity field itself is unknown right, and also the pressure field itself is unknown ok, that is what we are trying to solve from the given boundary conditions for the domain that we identified ok.

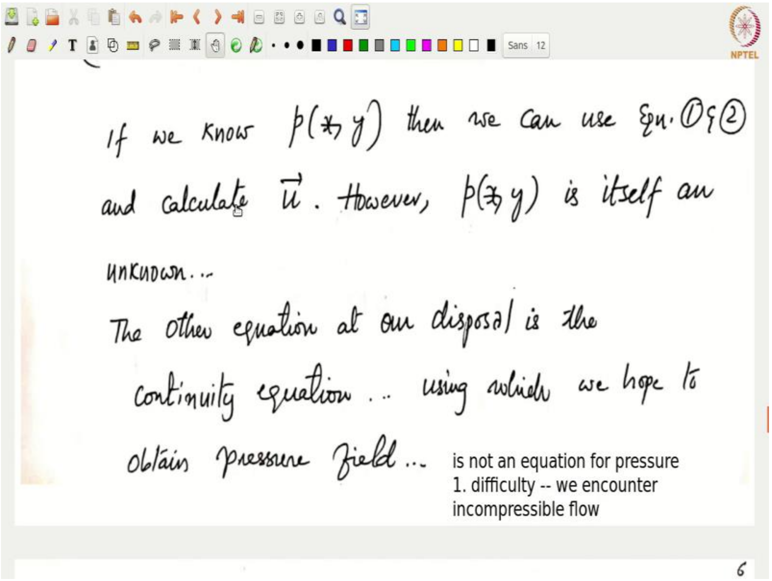
Now, because pressure at the faces is required, we cannot we do not know these values because pressure is assumed to be stored at the cell centers. So, we need a profile assumption for pressure. So, if we assume that the pressure varies linearly and we assume a uniform grid, then the pressure on east face can be written as p capital E plus p p by 2.

Similarly p little w would be written as P_W plus P_P by 2. And p on the north face can be written as P_N plus P_P by 2. And P on the south face can be written as P_S plus P_P by 2 ok. So, we essentially made a arithmetic average.

If we substitute these back into the this equation, then what we get is because P_P comes up in both equations and there is a minus here, we end with P_W minus P_E times Δy for the pressure gradient term in the x-momentum equation. And for the y-momentum equation, we end up with P_S minus P_N times Δx, so this is P_S minus P_N times Δx alright.

So, now we kind of translated the pressure gradient terms into cell pressures ok, from the face pressures to cell pressures is what we have done. So, these are now in terms of cell pressure values ok.

(Refer Slide Time: 32:29)



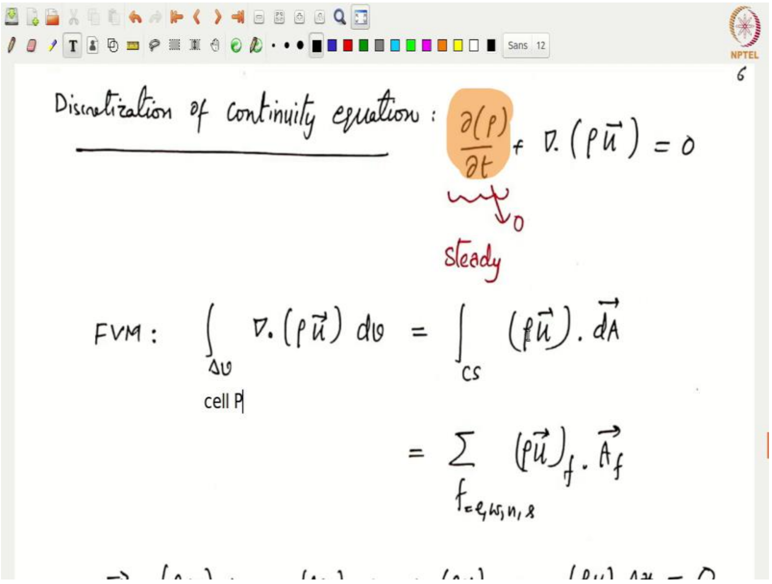
So, essentially if we know the pressure field that is we know the p x, y somebody gives you gives us, then we can use the equations above, and of course, calculate for the velocity field ok. But the pressure field itself is an unknown and the only equation that we have at our disposal is the continuity equation ok.

So, the continuity equation is the only one that we have remaining using which we hope to obtain the pressure field. However, we realize that the continuity equation is not an equation for pressure ok, so that is the first difficulty.

So, this difficulty 1 is basically we have three equations and three unknowns. However, the first equation is the x-momentum equation is an equation for u, the y-momentum is an equation for v. Whereas the equation for pressure is missing because the continuity equation is also in terms of velocities, but not in terms of pressure ok. So, this is the first difficulty we encounter when we try to solve a fluid flow equations for incompressible flow ok.

Because you remember if you if the flow is compressible, then we know that this is also a extra thing that is basically your density comes into play, and it through a equation of state the density and pressure are related. As a result things will be little more are actually balanced right which is not the case for incompressible flows because density is now constant ok, so that is the first difficulty. We will see what are the difficulties, we will get in order to solve the fluid flow equations for the incompressible range ok.

(Refer Slide Time: 33:57)

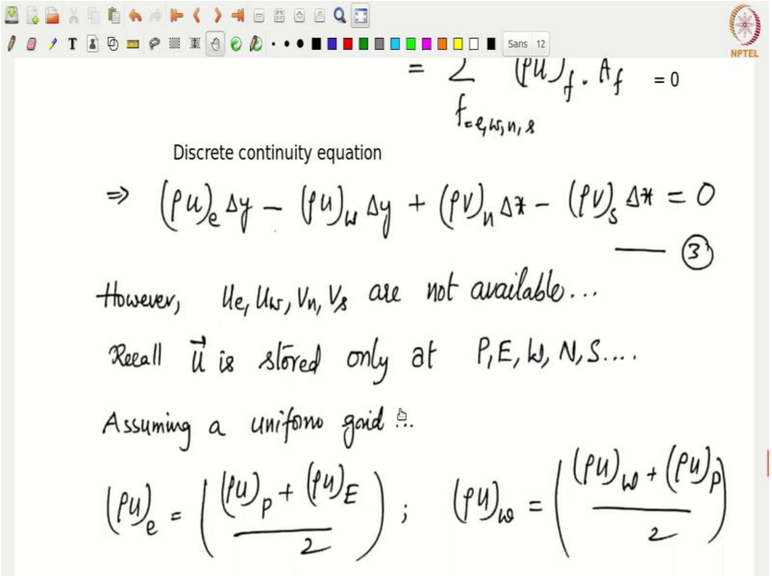


Then let us look at the discrete continuity equation. So, the continuity equation is basically $\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 00$, because we are assuming steady flow this particular term here is assumed to be this is basically goes to 0. So, we do not have $\frac{\partial \rho}{\partial t}$. So, only thing we have is $\nabla \cdot (\rho \vec{u}) = 0$ ok.

So, if we apply finite volume method, again we integrate this on the cell P that is integral ΔV this is integrated on cell P ok. So, $\nabla \cdot (\rho \vec{u})$ dV equals 0, that means, equals integral control surface applying invoking Gauss divergence theorem we can write this as $(\rho \vec{u}) \cdot$

$\overrightarrow{dA}$. Again assuming that $\rho\vec{u}$ is can be representing using the face value, and that is face centroid value is the representative for the entire face we can write this as $(\rho\vec{u})_f \cdot \overrightarrow{A_f}$ ok.

(Refer Slide Time: 35:04)



So, basically we have all the four faces – east, west, north, south. And this quantity would be equal to this is basically equal to 0 that is your discrete continuity equation ok. Now, we have four faces. So, we can expand this $\overrightarrow{A_f}$ is known A_e, A_w is basically known similarly A_n, A_s.

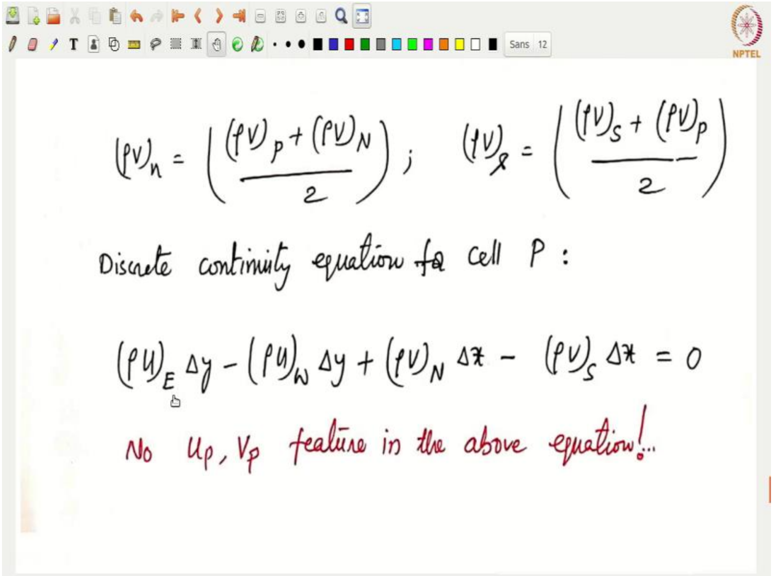
So, this can be extra expanded. And what we get is you get $(\rho u)_e \Delta y$ minus $(\rho u)_w \Delta y$ because $\overrightarrow{A_w}$ is $-\hat{\imath}\Delta y$, and similarly $(\rho v)_n \Delta x$ minus $(\rho v)_s \Delta x$. This is nothing but if you go back to the convection discussion this is nothing but F_e minus F_w plus F_n minus F_s right. So, this is basically conservation of mass or continuity equation. This should be equal to 0 right.

So, this is your discrete continuity equation right in terms of the face velocities. The velocities are now u_e, u_w, v_n and v_s which of course are not available right. Because, we are not we have not been storing velocities of the faces. We have been only storing velocities at the cell centroid that is velocity is only available at cell P centroid E, W, N and S all capital letters here.

So, if we assume uniform grid for the sake of simplicity, then we can write again using linear interpolation $(\rho u)_e$ as density is anyway constant. So, this we can write it as

$\left(\frac{(\rho u)_P+(\rho u)_E}{2}\right)$ ok. Similarly, $(\rho u)_w$ as $\left(\frac{(\rho u)_W+(\rho u)_P}{2}\right)$.

(Refer Slide Time: 36:42)



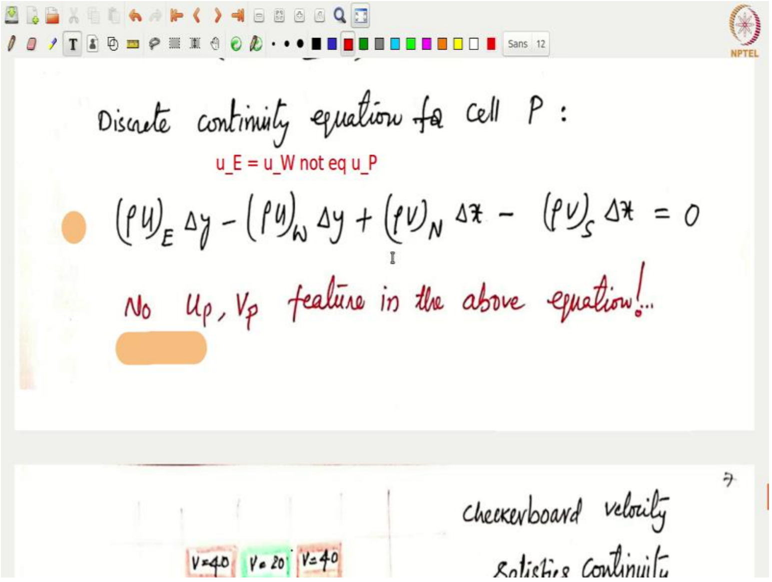
And similarly $(\rho v)_n$ can be written as $\left(\frac{(\rho v)_P+(\rho v)_N}{2}\right)$; and $(\rho v)_s$ can be written as $\left(\frac{(\rho v)_S+(\rho v)_P}{2}\right)$ ok. So, we have all these four values which we can substitute back here. And as you can see again similar to the pressure case the $(\rho u)_P$ gets cancelled with the $(\rho u)_P$ here, because we have a east minus west. So, this gets cancel.

As a result we get $(\rho u)_E \Delta y$ right, so discrete continuity equation of cell P would read as $(\rho u)_E \Delta y$ minus $(\rho u)_W \Delta y$ plus $(\rho v)_N \Delta x$ minus $(\rho v)_S \Delta x$ ok. So, this is your now discrete continuity equation for cell P ok.

Now, what we see here is that this is surprising because we have written equation for cell P, and none of the values of u_P or v_P which correspond to the cell P featuring the above equation. So, this equation is an equation for cell P, but the values of that particular cell are not there in the equation ok. So, that is something has very strong implications ok.

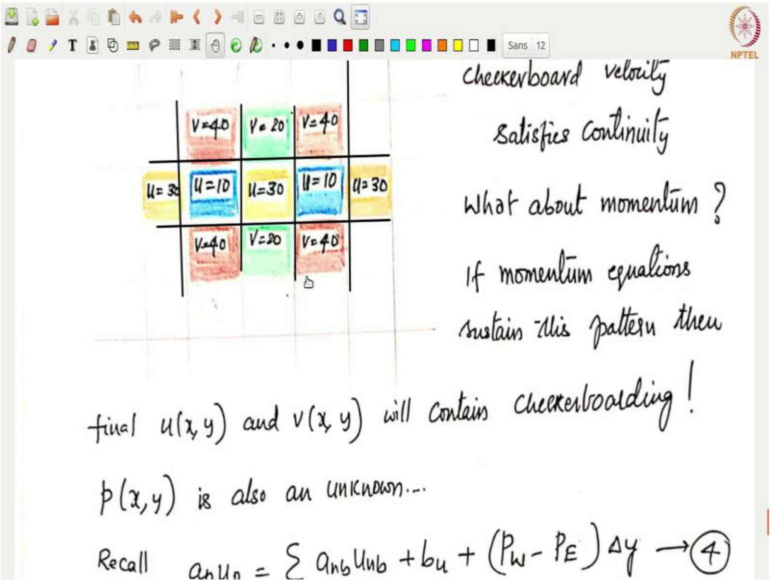
So, we will see what implication does it have because of this thing ok. So, essentially because of the u_P, v_P not featuring in the above equation, we will see that there is a major consequence of that a major trouble is kind of coming in the in the process of the solution procedure alright. That means, this kind of equation because your u_P, v_P do not feature in the equation, this can actually support a checker boarded kind of velocity pattern.

(Refer Slide Time: 38:26)



For example, if u east and u w are of the same value; however, they are different from the value of u_P ok. So, if u east equals u w, however they are that is not equal to u_P values ok. Similarly, v n equals v south, but they are not equal to v_P, then this equation can still be satisfied although there are oscillations in the velocity field this equation will still give you a conserved continuity equation. What that means, is that basically this kind of an equation supports checker boarding right.

(Refer Slide Time: 39:04)



So, for example, if you see this is basically these are different cells ok, each of the colors here is a cell. So, what we have is basically this is a particular cell u equal to 30 is a cell. Similarly, this is the grid I think the grid is quite light here you cannot see ok. So, that means, u lets say if this is your p cell p cell has u equals 30. However, as you can see from this equation only u east and u w matter.

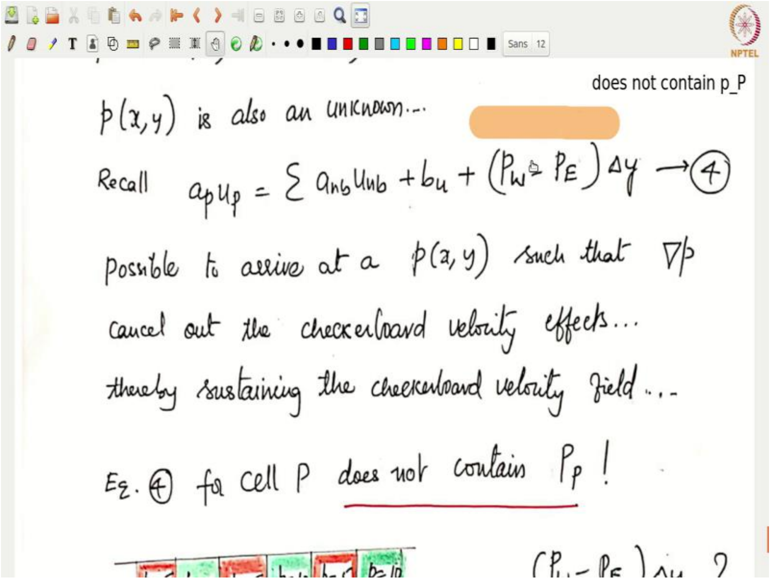
So, this is 10 and this is 10, so, this 10 minus this 10 that gets that satisfies. Similarly, v north and v south would be 20, 20. So, as a result this gives you perfect conservation of mass for this kind of a checker boarded velocity right. Although, we know that the velocity itself in the u and v components there are oscillations right, there is a kind of a chess board or a checker board kind of pattern existing which is satisfied by this kind of a continuity equation ah, but ok.

But we also know that we are not just solving for the continuity equation right. So, if continuity equation supports this kind of thing, let it be because we have the other equations which are the momentum equations. What about the momentum equation for u and v? They may not support this kind of a checker boarded velocity pattern. As a result this will be thrown out at some point in the solution procedure.

However, ok, but so, but if the momentum equation for some reason sustains this pattern right, then this will be; this will be contained, this will be there in the final solution as well ok. So, because this is supported by continuity equation and if the momentum equation sustain this pattern for some reason, then the final u and v will contain checker boarding.

We do not want checker boarding, because that is not something that is physical ok. The we do not want these kind of oscillations prevailing in the solution and still our solution kind of converges to some value ok. But we also recall that the pressure field itself is an unknown right, pressure field is an unknown.

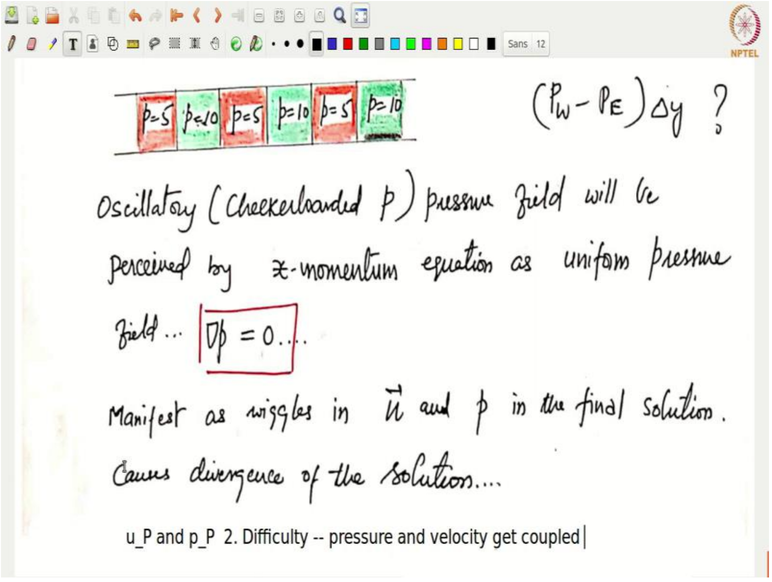
(Refer Slide Time: 41:10)



And if we go back to the kind of the x-momentum equation, we know that the discrete equation is $a_P u_P = \sum a_{nb} u_{nb} + b_u - (P_W - P_E)\Delta y$ ok. So, what we see here is that the pressure difference term, the pressure gradient term also this also does not this also does not contain P_P right.

Although this is an equation for cell P, P_P value does not feature here, that means, this also seems like it will support a checker boarded checker boarded pressure right. In which case if you have P_W equal to P_E, this would give a zero pressure gradient right, which is not true because P_W equals P_E, but neither of them would be equal to P_P ok.

(Refer Slide Time: 42:04)



So, that means, if you have so with such a pressure discretization, it is possible to arrive at a pressure field such that the gradient of p that is this term in the momentum equations might cancel out the effect that is coming because of the velocities itself because of the u_{nb}'s. As a result the velocity checker boarding could be sustained by the momentum equation as well ok.

So, what we are talking about is basically you have the pressure field, the pressure field might have a checker boarded pattern like this essentially we have p equal to 5, p equal to 10, p equal to 5, p equal to 10 and 5 and so on. So, essentially we have oscillatory field which will be seen by the x-momentum equation as what as a zero gradient pressure field. Because if you look at particular p, this is 5 minus 5, so this would be basically zero pressure gradient.

So, this is a zero pressure gradient. As a result you would see that this momentum equation cannot see this pressure gradient ok. So, such a pressure boarding pressure checker boarding is now not seen by the velocity by the momentum equation ok. Similarly, there could be some pattern of pressure field which might cancel out those pressure gradients here might cancel out a velocity field like this.

That means, this negated with a pressure checker boarding could give you a free run for your velocity and the pressure fields, such that they contain these kind of an oscillatory field in the final solution ok. Do you see that?

So, the pressure oscillations could the pressure oscillations the gradient of which could cancel out the velocity checker boarding as a which will in term sustain the checker boarded velocity field not only in the continuity equation, but also in the momentum equations because now your pressure field might support it ok. So, you may get such a pressure gradient field which might cancel out the velocity checker boarding ok.

As a result, if this happens usually if this happens when you try to solve it and as a result the oscillatory essentially you your final solution will have wiggles in the velocities and pressures. And these wiggles will manifest themselves into the divergence of the solution ok. So, that means, what we have seen is the oscillatory pressure field will be perceived by x-momentum equation as uniform pressure field right. The gradient of p is 0.

Now, there could be some other pressure field which might cancel out the effects created by a checker boarded velocity field ok, which might basically help sustain this checker boarded velocity field through the momentum equation as well as of course through the continuity equations ok.

As a result there might be some wiggles that might start showing up, these, these are the wiggles in the pressure and the velocities which would eventually cause divergence in the solution, these kind of oscillatory behaviour. And, that is exactly what happens in when you try to solve it the incompressible fluid flow equations when you store all the velocities and pressures at the cell centroids ok.

So, this is the second difficulty that is basically the pressure and velocity get kind of they kind of get coupled. And in a sense they kind of support this kind of a checker boarded patterns in velocities and pressures which will manifest into divergence of the solution eventually ok. So, this is the second difficulty. The first difficulty we saw was that there is no equation for pressure.

The second difficulty is that the pressure and velocity because of their coupling, they may together they may support a checker boarded velocity and pressure patterns in the final solution ok, so that is the other difficulty. So, we will see how to address these two difficulties when we try to solve a fluid flow equations through some algorithms that are there in the literature. So, that will be our topic for another lecture on another day ok.

So, I am going to stop here. If you have any questions, let me know through e mail alright.

Talk to you in the next lecture.

Thank you.