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

Lecture - 24
A Finite Volume Method to solve Three-dimensional NS Equations in Complex Geometry

(Refer Slide Time: 00:34)

Semi-Explicit Time-Stepping for the Navier-Stokes Equations

Subtrcting equation (41) from (39), we get

$$\rho V \frac{\boldsymbol{u}'_P}{\Delta t} = -\sum_j p'_j \boldsymbol{S}_{ij} \qquad (42)$$

where the corrections

$$\boldsymbol{u}'_P = \boldsymbol{u}_P^{n+1} - \boldsymbol{u}_P^* \ , \qquad p'_j = p_j^{n+1} - p_j^n \qquad (43)$$

and the corresponding flux corrections F'_j have to satisfy

$$\sum_j F'_j = -\sum_j F^* \qquad (44)$$

Good morning, everybody. Today, we will discuss the concluding part of Finite Volume Method to solve Three-dimensional Navier-Stokes Equations in Complex Geometry. If we recall in the last class, we discussed about equation 42. Equation 42 is basically, outcome of equation 41 and 39.

39 and 41 both are momentum equations. 39 pressure is at level n+1 and then the velocities are advanced at the level n+1 and we can call all the velocities are at a level n+1, because the pressure is also at a level n+1. This is basically the projected velocity.

In calculation what we do? We do not get pressure at the n+1th level. We get a level at level n, and then through the convection diffusion equation and pressure contribution, we get a velocity, which is provisional velocity, and this velocity has not yet satisfied the continuity equation in each cell.

So, these velocity quantities in each cell, u v, and w, are provisional velocities, and there is a difference between this provisional velocity and the final velocity u v w, n+1, and this difference we can call the velocity correction. If we recall, in Mach or simple algorithm, we have used a basically superscript see; here, we have used prime.

So, this is velocity correction and similarly pressure at a level n+1 minus pressure at a level n, if we subtract, we get p'. This is also synonymous to pressure correction, then we can you know, basically what we said has been written here that $\boldsymbol{u}_p'$ is $\boldsymbol{u}_p^{n+1}$ minus $\boldsymbol{u}_p'$. These **u**'s are all vector u; that means, they signify u v w.

Then p' is p^{n+1}-p^n at jth cell or j face and then the corresponding flux corrections so, since the provisional velocities if we use then the fluxes; that means, the mass flux will not satisfy continuity equation. So, we require to correct it and the corresponding flux corrections will be then on all confining surfaces j summed over east, west, north, south, top and bottom side of the cell equal to minus then again j summed over all confining surfaces F^*.

(Refer Slide Time: 04:42)

Usual Pressure-Velocity Correction Method

Equation (42) and (44) can be simultaneously solved by the following method:

1. Assume some guesses for the cell-center pressure corrections $p_P^{'}$.
2. Use interpolation to obtain the corresponding face-center correction $p_j^{'}$
3. Use Eqn. (42) to get cell-center velocity correction $\boldsymbol{u}_P^{'}$
4. Use interpolation to get the face-center correction $u_j^{'}$ and compute the convection fluxes $F_j^{'}$.

$$F_j^{'} = \rho \boldsymbol{u}_j^{'} \cdot \boldsymbol{S}_j$$

F* is basically that uses u*s. So, this is equation 44. Now, equation 42 and 44 can be simultaneously solved by the following method. These we have already discussed. Assume some guesses for the cell center pressure corrections which is p_P', P ,here the suffix P is a signifying center of the cell.

Use interpolation to obtain the corresponding face center corrections; that means, the corrections that are applicable to each face then and the center of the cell it is p_P' at each phase p_j'. Use equation 42; that means this equation, equation 42 to get cell center velocity correction u_P'.

Use interpolation to get the phase center correction u_j' from u_P' and compute the convection fluxes which is F_j'. So, F_j' equal to then $\rho \boldsymbol{u}_j' . \boldsymbol{S}_j$. **S** is a surface vector of each confining surface and $\boldsymbol{u}$ is the vector quantity means the velocities in x, y and z directions.

(Refer Slide Time: 06:35)

Usual Pressure-Velocity Correction Methods

5. Compute the residual (difference between the RHS and LHS) of Eq. (44).
6. We can use this residual to get new values of corrections p_P' (An improved and faster way of doing this involves the *conjugate gradient method*) and repeat steps 2, 3, 4, 5 and 6 till the residuals are zero (i.e. less than a pre-specified small value). Care has to be taken that the interpolation in steps (2) and (4) are done in a manner such that slow convergence does not result.
7. Experience suggests that form of momentum interpolation may have to be used in step (4), instead of essentially the linear interpolation proposed earlier.

Now, compute the residual; that means, right hand side and the left-hand side of equation 44 which is again equation 44, let us recall. So, basically this will be for a converse situation, this will produce zero. Now, RHS and LHS of equation 44, we can use the residual to get new values of corrections p_P' prime an improved and faster way of doing this involves when we conjugate gradient method and repeat steps 2, 3, 4, 5 and 6 till residuals are zero, that is and in computation getting exactly zero is not possible.

So, we will define a basically very small number a predefined upper bound and pre specified small value. Care has to be taken that the interpolation in steps 2 and 4 are done in a manner. So, again let us look at step 2 and step 4. So, these interpolations are to be done in a manner such that slow convergence does not result. Experience suggests that from the momentum interpolation, I mean form of momentum interpolation may have to be used in step 4 instead of essentially linear interpolation.

(Refer Slide Time: 08:45)

Modified Pressure-Velocity Correction Method

If the discretised equation for velocity is written as

$$\boldsymbol{u}_P^* = \boldsymbol{u}_P^{old} - \frac{\Delta t}{\rho V_P}(F_P^c + F_P^d) + \frac{\Delta t}{\rho V_P} S_u \quad \textbf{(45)}$$

where $\boldsymbol{u}_P^{old}$ is the n^{th} time-step velocity and S_u is the pressure term. We define a new variable $\boldsymbol{v}_P$ which is

$$\boldsymbol{v}_P = \boldsymbol{u}_P^{old} - \frac{\Delta t}{\rho V_P}(F_P^c + F_P^d) \quad \textbf{(46)}$$

Now to estimate the mass flux $F_e = \rho \boldsymbol{u}_e \cdot \boldsymbol{S}_e$ at the east face, straight forward linear interpolation would use

$$F_e = \rho(\overline{\boldsymbol{u}_P, \boldsymbol{u}_E}) \cdot \boldsymbol{S}_e \quad \textbf{(47)}$$

So, step 4 we may deploy momentum interpolation technique. How to do that? That is what is our modified pressure velocity correction method. Now, this step 1 to 6 what we have written this is working algorithm and we want to make it little more robust, little faster. So, now, we go for modified pressure velocity correction method. If the discretized equation for velocity is written as u_P^* equal to u_P^{old} minus Δt divided by ρV_P. This upper-case V is a, we have already seen this is a volume of the cell. So, cell P, ρV_P and these are basically convection and diffusion contributions. Then again Δt, from lift hand side Δt has come to this side divided by rho volume of the cell P equations 45

$$\left(u_P^* = u_P^{old} - \frac{\Delta t}{\rho V_P}(F_P^c + F_P^d) + \frac{\Delta t}{\rho V_P} S_u\right) \quad (45)$$

We have to remember equation 45 throughout in order to, you know properly understand the modified pressure velocity correction method. Now, u_P^{old} this is; obviously, the nth time step velocity, nth u_P^{old} means u_P^n and S_u is a pressure term. We define now, we if we can ignore this part, this is a velocity that we are defining.

This is you know sort of, we have not worked with it so far. So, we define a new variable V_P again, vector quantity at point P of the cell P and this as also 3 components, but this excludes pressure contribution. Many books it is written as mass velocity so, but its matter of definition we will understand it by the fact that there is no pressure contribution and this is we will call V_P. Now, to estimate the mass flux F_e we will write ρu_E; that means, at the eastern face center, $\boldsymbol{S_e}$ at the east face.

Straight forward linear interpolation when we have done, I mean when we apply linear interpolation basically, you know the eastern neighbor and the cell of interest P and E they get involved and do interpolation between them linear interpolation. We have already done it you know in our second lecture, in our first lecture and second lecture both we have discussed that and we will mention it here again. This is given by equation 47.

(Refer Slide Time: 12:29)

Modified Pressure Velocity Correction Method

where the overbar indicates a linear interpolation using Eq. (16). But momentum interpolation would use

$$F_e = \rho\,(\overline{\boldsymbol{v}_P, \boldsymbol{v}_E}) \cdot \boldsymbol{S}_e - \Delta t\, \boldsymbol{\nabla} p \cdot \boldsymbol{S}_e \qquad \textbf{(48)}$$

where the gradient ∇p is estimated using p_P, p_E and also other neighbours for a non-orthogonal grid. The role of successive under-relaxation with non-staggered grids has been investigated by Majumdar (1998).

$$\phi_e = \frac{V_E}{V_E + V_P}\phi_P + \frac{V_P}{V_E + V_P}\phi_E \qquad (16)$$

Where the over bar so, basically that linear interpolation is indicated by this over bar. Over bar symbol means that linear interpolation which we used as equation 16 and what was it? If we recall that at eastern face center the quantity ϕ will be this is volume of eastern face, eastern cell, eastern neighbor divided by volume of eastern neighbor plus volume of the cell of interest into ϕ_P and the center of the cell P plus volume of cell P divided by volume of eastern neighbors' volume of cell P total value into ϕ_E.

$$\phi_e = \frac{V_E}{V_E + V_P}\phi_P + \frac{V_P}{V_E + V_P}\phi_E \qquad (16)$$

So, this was a linear interpolation pressure ϕ_E into this factor ϕ_P into this factor ϕ_P is at the center of cell P, whose volume as V_P, ϕ_E is a center of cell E whose volume is V_E and this interpolation, linear interpolation kind of central difference gives us ϕ_E at the face center.

So, now what we do if we look at a here that F_e has to be calculated and we have done it straight forward through linear interpolation earlier and now what we will do? We will do momentum interpolation which means we will do linear interpolation of these velocities which do not have contribution of pressure this V_P and V_E.

So, linear interpolation on these 2 quantities for eastern cell face and contribution of pressure and this contribution of pressure is basically the gradient ∇p is estimated using pressure of at the point P pressure at the point E and also other neighbours for a non-orthogonal grid the role of and this is sometimes this contribution also under relaxed and the role of this under relaxation with non-staggered grid, because this is collocated grid arrangement has been investigated by Majumdar 1998.

(Refer Slide Time: 15:34)

Modified Pressure-Velocity Correction Method
Solution Algorithm

The velocity and pressure fields are calculated with the following Gauss-Seidel type algorithm (Eswaran and Prakash, 1998):

1. Use Eq. (45) and (46) to compute the cell-center $\boldsymbol{u}_P$ and $\boldsymbol{v}_P$. Make an initial guess p_P. Use interpolation Eq. (16) of earlier slide to obtain the face-centered quantities $\boldsymbol{v}_j$ and p_j.
2. Compute the mass flux through each face j using
$$F_j^* = \rho \boldsymbol{v}_j \cdot \boldsymbol{S}_j - \Delta t\, \boldsymbol{\nabla} p_j \cdot \boldsymbol{S}_j \quad \textbf{(49)}$$
3. Use equation $\boldsymbol{u}_j' = -\frac{\Delta t}{\rho} \boldsymbol{\nabla} p_j'$ to compute the flux correction at the face j
$$F_j' = \rho \boldsymbol{u}_j' \cdot \boldsymbol{S}_j \text{ or } F_j' = -\Delta t\, \boldsymbol{\nabla} p_j' \cdot \boldsymbol{S}_j \quad \textbf{(50)}$$

It is a very important paper. I would suggest all of you to read this paper. I will give the reference of this paper at the end of the lecture. The velocity and pressure fields are calculated. So, this new algorithm that Gauss-Seidel type algorithm of Eswaran and Prakash that we have already a seen, but this modified algorithm what can be done, but this algorithm is also proposed by them and here also Gauss-Seidel type iterations are used. So, use equation 45 and 46, I suggested earlier 45 and 46.

So, basically 45 and 46 to compute cell center u_P and v_P make an initial guess of p_P pressure and the cell of interest. Use interpolation of equation 16. Equation 16 means linear interpolation of earlier slide; that means, this we have written in this here by equation 16 to obtain face centered quantities v_j and p_j.

These are true linear interpolation. Compute mass flux through east face j by using this. So, this mass flux F_j^* star is $\rho \boldsymbol{v}_j . \boldsymbol{S}_j$ - $\Delta t \boldsymbol{\nabla} p_j . \boldsymbol{S}_j$. Use equation u_j' is $(-\Delta t$ by $\rho)$ $\nabla p_j'$ to compute flux correction at face j, then this u_j' is used for as a as has been written here flux correction. So, if we substitute $\left(-\frac{\Delta t}{\rho}\right)\ \nabla p'_j$ for u'_j we get this expression which is equation 50.

(Refer Slide Time: 18:20)

Modified Pressure-Velocity Correction Method
Solution Algorithm

4. Compute the residual for each cell

$$\Re = -\sum_j F_j^* - \sum_j F_j' \qquad \textbf{(51)}$$

5. Calculate the cell-center pressure correction from the relation

$$p_P' = p_P' + \omega \frac{\Re}{\alpha_P} \qquad \textbf{(52)}$$

This is computed using the formulation for diffusion fluxes (Eq. (24)-(33)) replacing ϕ by p'. Here ω is the relaxation factor. If $0 < \omega < 1$, it is called *underrelaxation, and for* $\omega > 1$, it is *overrelaxation*. Again, α_P stand for diagonal coefficients that can be calculated using the equation below

$$\alpha_P = \Delta t\left[\frac{\alpha_1}{\Delta x^1}\Big|_w - \frac{\alpha_1}{\Delta x^1}\Big|_e + \frac{\alpha_2}{\Delta x^2}\Big|_n - \frac{\alpha_2}{\Delta x^2}\Big|_s + \frac{\alpha_3}{\Delta x^3}\Big|_b - \frac{\alpha_3}{\Delta x^3}\Big|_t\right] \qquad \textbf{(53)}$$

where α_1, α_2 and α_3 are the same as in Eqs. (24-33).

Now, in each cell we compute this residual which is basically minus this provisional velocity minus pressure, a flux velocity correction flux and then calculate cell center pressure correction from the relation p_P' is $p_P' + \omega \frac{\Re}{\alpha_P}$. This is computed using the formulation for diffusion fluxes replacing phi by p' .

So, this p' just the way we diffusion flux calculated diffusion flux, we will use the same technique and we have to mention this omega, lower case omega is a relaxation factor. If omega is between 0 and 1 it is called under relaxation; that means, less than 1 and for omega greater than 1 is over relaxation.

And α_P stands for diagonal coefficient that can be calculated by using

$$\alpha_P = \Delta t \left[\frac{\alpha_1}{\Delta x^1}\bigg|_w - \frac{\alpha_1}{\Delta x^1}\bigg|_e + \frac{\alpha_2}{\Delta x^2}\bigg|_n - \frac{\alpha_2}{\Delta x^2}\bigg|_s + \frac{\alpha_3}{\Delta x^3}\bigg|_b - \frac{\alpha_3}{\Delta x^3}\bigg|_t \right] \quad \textbf{(53)}$$

So, this is there is a small mistake; please note it down equation 53 here. This is not minus; this should have been plus. Please, note down this mistake, this has to be plus. So, where α_1, α_2, α_3 are basically the same parameters that we calculated in our previous lecture from the basically, components of unit normal vector n1 in 3 Cartesian direction, n2 in 3 Cartesian direction, and n3 in 3 Cartesian direction.

And from there, we got basically coefficient matrix multiplied by α_1, α_2, α_3 equal to S_{1j} S_{2j} S_{3j} from their determinants d of the coefficient matrix and replacing one the columns by S_{1j} S_{2j} S_{3j}, we got d_1 d_2 d_3. So, α was α_1 was d1 by d when in the co-efficient matrix first column is replaced by S_{1j} S_{2j} S_{3j} we get d_1 and d_1 by the determinant of d gives α_1. Similarly, d_2 by d gives α_2 and d_3 by d determinant, the coefficient matrix d gives α_3.

(Refer Slide Time: 22:21)

Modified Pressure-Velocity Correction Method
Solution Algorithm

6. If $\Re_{rms} > \epsilon$, go to step (3).
7. Store the updated mass flux through cell-faces from

$$F_j^* = F_j^* - \Delta t\, \nabla p_P' \cdot \boldsymbol{S}_j \quad \textbf{(54)}$$

8. Store the updated pressure at cell-center $p_P = p_P + p_P'$.
9. Store the cell-centered corrected velocity

$$\boldsymbol{u}_P' = -\frac{\Delta t}{\rho_P V_P} \sum_j p_j' S_{ij} \quad \textbf{(55)}$$

$$\boldsymbol{u}_P = \boldsymbol{u}_P + \boldsymbol{u}_P' \quad \textbf{(56)}$$

Now, this a residue, we calculate the rms value. If it is less than epsilon it has already converge, if it is greater than epsilon then go to step 3 and repeat it again. So, store the updated mass flux through cell faces. This is we know this will be the updated mass flux and store the updated pressure, it will be updating updated pressure and store the cell centered corrected velocity and cell-centered corrected velocity is this and then you update the velocity by this velocity correction.

(Refer Slide Time: 23:27)

Modified Pressure-Velocity Correction Method

Solution Algorithm

10. Go to step (1) and repeat the process until steady state or dynamic steady state (for unsteady flows) is reached.

It can be shown that satisfying Eq. (44) is same as solving

$$\nabla^2 p' = -\frac{\boldsymbol{\nabla} \cdot \boldsymbol{u}^*}{\Delta t} \quad \textbf{(57)}$$

and finding the corrections

$$\boldsymbol{u}' = -\Delta t \nabla p' \quad \textbf{(58)}$$

Next, go to step 1 and repeat the process until steady state or dynamic steady state is reached. So, this the residual value, rms value will be 0 if we reach steady state. If we do not reach steady state for unsteady flow usually for an unsteady flow the velocities, will you know keep on varying maybe in keep on repeating in a cyclic manner or periodic manner. So, under such a situation we have to reach dynamic steady state. So, either we reach steady state or we reach dynamic steady state.

It can be shown that satisfying equation 44; that means, this summation of the mass flux correction equal to basically summation of F^* overall surfaces that will be you know basically, it closed to 0. as I said that I, 0 means steady state, but in numerical computation getting 0 is difficult. So, it will be less than some predefined upper bound maybe 10 to the power minus 3 or 10 to the power minus 4 then we have reached the steady state.

So, it can be shown that an equation 44 satisfying, equation 44 is basically equivalent to solving

$$\nabla^2 p' = \frac{\nabla . u^*}{\Delta t}$$

when u^* reaches n+1 this divergence of u^{n+1} is 0. So, there will be no longer pressure correction.

Otherwise, equation 57 is basically Poisson equation for pressure correction. So, and simultaneously, we get velocity correction term when we get a non-zero pressure correction term, we get a velocity correction by done by equation 58 and basically, we keep on updating u^* through velocity correction so that u^{n+1} evolves and pressure also is modified through pressure correction so that pressure at the level n+1 evolves.

(Refer Slide Time: 26:30)

Modified Pressure-Velocity Correction Method

Solution Algorithm

such that

$$\boldsymbol{u}^{n+1} = \boldsymbol{u}^* + \boldsymbol{u}' \quad \textbf{(59)}$$

$$p^{n+1} = p^n + p' \quad \textbf{(60)}$$

This procedure above is equivalent to solution of the Poisson equation for the pressure correction.

After evaluating the correct velocities, the energy equation is solved with a *Successive Over-Relaxation* technique to determine the temperature field. As mentioned earlier, the numerical procedure of Eswaran and Prakash (1998) was used and the detailed calculations were outlined by Prabhakar *et al.* (2002). The method has been successfully applied to solve heat transfer problems (Tiwari *et al.* 2003a and 2003b) in complex geometry.

Dalal *et al.* (2008) extended this method for solving Navier-Stokes Equations using unstructured grid meshes.

The procedure which we have explained here is equivalent to solution of Poisson equation for pressure connection, we have already mentioned that. So, after evaluating the corrected velocities and corrected pressure the final pressure and final velocities, the energy equation is solved with the successive over relaxation technique to determine the temperature field. How it can be done while discussing Mach algorithm we have discussed.

See, basically after solving the velocity field when we go to energy equation no longer energy equation is non-linear, because velocity quantities in the convective term unknown.

So, in our solution is relatively easy and it can be and done through successive over relaxation technique.

The numerical procedure of Eswaran and Prakash was used and the detailed calculations were outlined by Prabhakar et al. in another investigation. It was very successfully deployed. Later on, it was again very successfully deployed by Tiwari et al. We can see Tiwari and coworkers to publication so, will mention in bibliography or list of references in very complex geometry. basically, the fluid flow problem was solved and then a heat transfer problem was also solved.

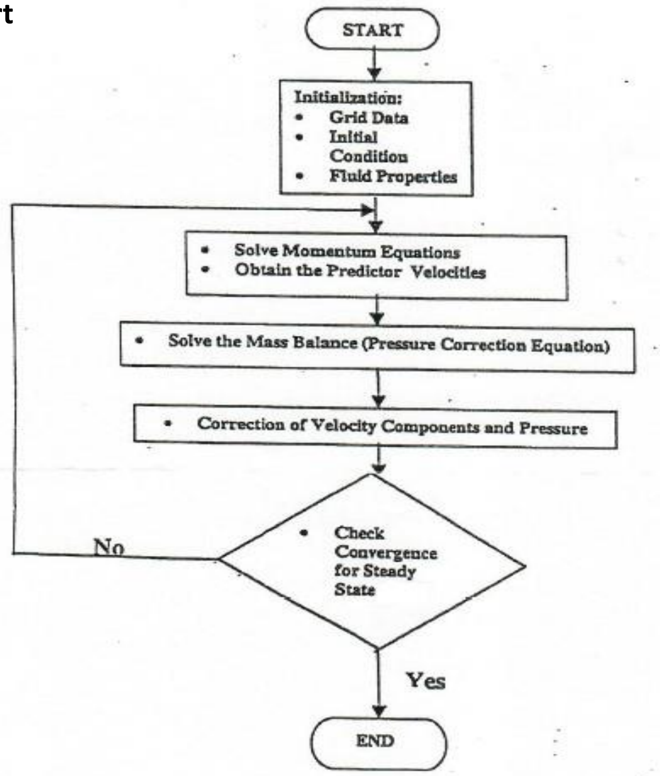
Dalal et al in 2008, extended this method for solving Navier-Stokes equations using unstructured grid meshes. So, a basically this is yet other technique. So, hexahedral when we use, even though these hexahedral can be of various orientation, various sizes, but they are count as i j k are structured.

Now, more complex geometries sometimes it is needed to adapt unstructured grids, because un structured grids deploy instead of hexahedral, tetrahedral and this tetrahedrons can capture the tortuous geometric tortuosity more accurately than hexahedrons.

So, for some geometries it is needed it is desirable to use tetrahedrons and when we use tetrahedrons, Basically, it becomes a paradigm of unstructured grids. So, solution methodology what we have outlined requires to be modified and that was done by Dalal et al.

(Refer Slide Time: 30:58)

Flow Chart



Now, I will briefly discuss about the flow chart. As you can see grid data grid has to be generated for meshing the geometry of interest and all x, y, z locations of all the points. They will finally, describe hexahedrons of various sizes, of various sizes and shapes, but these hexahedrons of various sizes and shapes are the grid.

They combined together grid, but the x, y, z coordinates that is a big data that is to be read. So, that conceptually all this hexahedral of you know as I said various shapes and sizes are conceptually described and we can apply boundary conditions suitably, then initial conditions are set and fluid flowing properties are; obviously, needed. We solve momentum equation to obtain basically, provisionally calculated velocities. Predictor velocity is meaning provisionally calculated velocities.

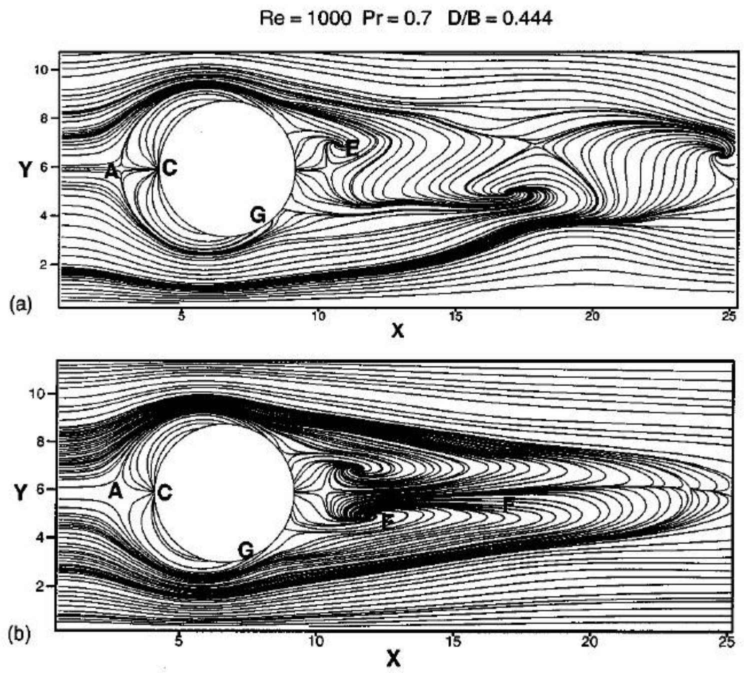
Then we solve the mass balance and basically try to comply with zero mass divergence in each cell and basically the also we correct the pressure so that we evolve from nth level to $n+1^{th}$ level with correct pressure and correct velocities at a level n+1, these corrections sorry, here solve the mass balance try to comply with zero mass divergence in each cell and it results in finally, correction in the velocity and pressure in each cell velocity components and pressure in each cell.

So, basically there is a loop here, there is we like outer loop, there is an inner loop and we have to keep on correcting velocity and pressure till zero mass divergence, which we indicated by equation 44, is evolved, and then we progress from level n to n+1 and here check the convergence for steady state, if it as reach steady state you know we have obtained the final velocity and the pressure field final velocities in the field and the pressure, if it as not converged we will repeat the calculation.

And having obtained the steady state velocity field or, you know, dynamically steady velocity field, then we have to calculate for unsteady flows, then we have to calculate the average velocities and with the either for with that average velocities or with the final steady state velocities we have to go for solving energy equation for finding out the temperature field in the domain. Now, I will show some sample calculations. This is a reasonably complex geometry, a cylinder circular cylinder in a channel.

(Refer Slide Time: 34:51)

Limiting streamlines describe the topological properties inferred from computed streamlines. According to **Legendre**, the topology of three-dimensional streamlines projected on a no-slip body surface is analogous to the experimental surface flow visualization. The skin-friction lines, according to **Lighthill**, can be regarded as the projection of the three-dimensional Streamline field on the body surface.



Limiting streamlines of the flow past a built-in circular tube in a channel at the bottom plate corresponding to (*a*) instantaneous flow field and (*b*) time-averaged flow field

It is not a 2-dimensional flow, 3-dimensional flow of a circular flow faster circular cylinder cross flow basically in a channel, Reynolds number 1000 and here what we have shown is little unfamiliar too many, these are called limiting streamlines. So, limiting streamlines means projection of stream lines on a surface.

So, basically this is on the bottom surface and what is limiting streamline and what is its importance? Here, I have written it here and you can see not only the (Refer Time: 35:54) the creators of the fields you have to say Legendre and Lighthill. You can get their views about this limiting streamline.

It is there is a special technique to plot the limiting streamlines. As you can see that according to Legendre the topology of 3 dimensional streamlines projected on a no slip body surface is analogous to experimental flow visualization and then Lighthill also supported this argument and these are very strong analytical tools for no analyzing nuances of flow or things like saddle points, things like you know nodes etcetera in analytical sense can be determined to this.

(Refer Slide Time: 37:43)

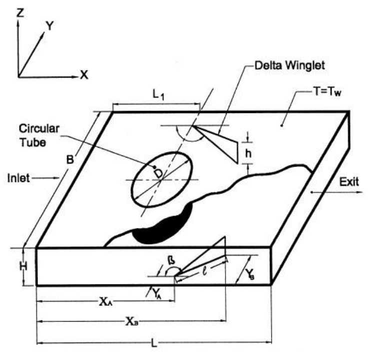
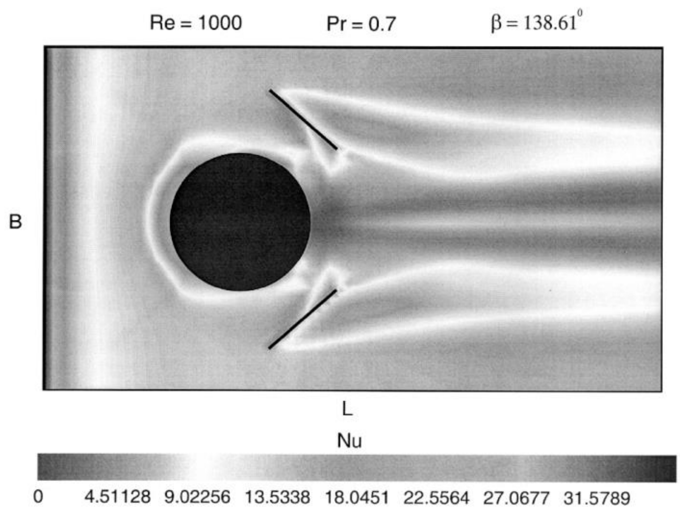


Figure 2. Heat exchanger module considered in the present investigation.



Nusselt number distribution on the bottom plate of a channel with built-in circular tube and delta winglet pair in common-flow-up configuration.

Here you can see basically instantaneous limiting streamlines on the bottom plane and this is unsteady flow so; obviously, you know it will vary from time to time and then if we plot the time average limiting stream lines, they we look like and this. We will go for another example this is taken from a flow simulation in a module of a fin tube heat exchanger. In fin tube heat exchangers see, this is a tube and there will be number of tubes through the tubes basically hot fluid flows.

And it is cooled by the cooler fluid stream and these tubes are surrounded by fins. So, this top plate is a fin maybe another tube is sitting here, maybe another tube sitting here so and bottom plate is also fin. So, these tubes are surrounded by fins and through this the gap which we can see rather throughout through all the gaps cooler streams flow to basically cool the heated fins and the tube through which hot fluid is flowing, this is another fluid.

So, and these flowing fluids what we are considering now is maybe, maybe water a cooling fluid and we can see a pair of Delta Winglet are Winglets are sitting behind., this Delta Winglets are also known as vortex generators. So, these vortex generators generate vortices and enhance the heat transfer.

Now, here on the bottom plate the a Nusselt number distribution which is index of a transfer has been plotted and basically the this is color code, but we have used you know converted to the colored picture into black and white, but you can see the variation of Nusselt number on this scale and this grid scale also the variation of Nusselt number is indicated on the bottom surface. This is the location of the tube and this is the location of the Delta Winglet pair.

(Refer Slide Time: 40:25)

V. Eswaran and S. Prakash, A Finite Volume Method for Navier-Stokes Equations, Proceedings of the Third Asian CFD Conference, Bangalore, Vol. 1, pp. 127-133, 1998

W. Kordulla and M. Vinokur, "Efficient Computation of Volume in Flow Predictions" *AIAA Journal,* Vol 21, No. 6, pp. 917-918, 1983.

P.K. Khosla and S.G. Rubin, A Diagonally Dominant Second-Order Accurate Implicit Scheme, *Computers and Fluids*, Vol. 2, pp. 207-209, 1974

S. Majumdar, Role of underrelaxation in momentum interpolation for calculation of flow with non-staggered grids, *Numerical Heat Transfer*, Vol. 13, pp. pp. 128-132, 1988

V. Prabhakar, G. Biswas and V. Eswaran, Numerical Prediction of Heat Transfer in a Channel with Built-in Oval Tube and Two Different Shaped Vortex Generators, *Numerical Heat Transfer, Part A*, Vol. 41, pp. 307-329, 2002

S. Tiwari, G. Biswas, P.L.N. Prasad and S. Basu, Numerical Prediction of Flow and Heat transfer in a Rectangular Channel with a Built-in Circular Tube, *Journal of Heat Transfer (ASME)*, Vol. 125, pp. 413-421, 2003

S. Tiwari, D. Maurya, G. Biswas and V. Eswaran, Heat Transfer Enhancement in Crossflow Heat Exchangers using Oval Tubes and Multiple Delta Winglets, *International Journal of Heat and Mass Transfer*, Vol. 46, pp. 2841-2856, 2003

A. Dalal, V. Eswaran and G. Biswas, A Finite Volume Method for Navier-Stokes Equations on Unstructured Meshes, *Numerical Heat Transfer, Part-B*, Vol. 54, pp. 238-259, 2008

Now, having done that we will a mention about the references. As I have already said the basic contribution of Professor Eswaran and Doctor Satya Prakash and other important papers like a Professor Kordulla paper, Professor Khosla, and Professor Rubins paper, Professor Sekhar Majumdars paper.

And all other important papers have been referred here. You can consult these papers for your own benefit. And with this we have come to the end of our discussion on basically the complex geometry and the fluid flow and heat transfer analysis in complex geometry. We have primarily done fluid flow analysis and we have been exposed to very novel algorithm, very powerful algorithm. I would encourage all of you to take up any geometry that you think you would be able to handle.

And try to make use of this algorithm. Try to come up with your own program, with your own code and get the results and I am hopeful that many of you would be you know working on development of codes for them this knowledge is very important, but people who would be using codes.

That are available commercially available codes or open-source codes for them also knowing the techniques, what is to be done for basically you know at different stages for implementation of basically different schemes of different orders or basically how to calculate a convective flux, how to calculate diffusive flux, how to correct pressure, these are indeed very important knowledge. I am thankful to you for your interest., I hope your interest will keep on increasing thank you very much. Hopefully, we will interact again, we will meet again.

Thank you.