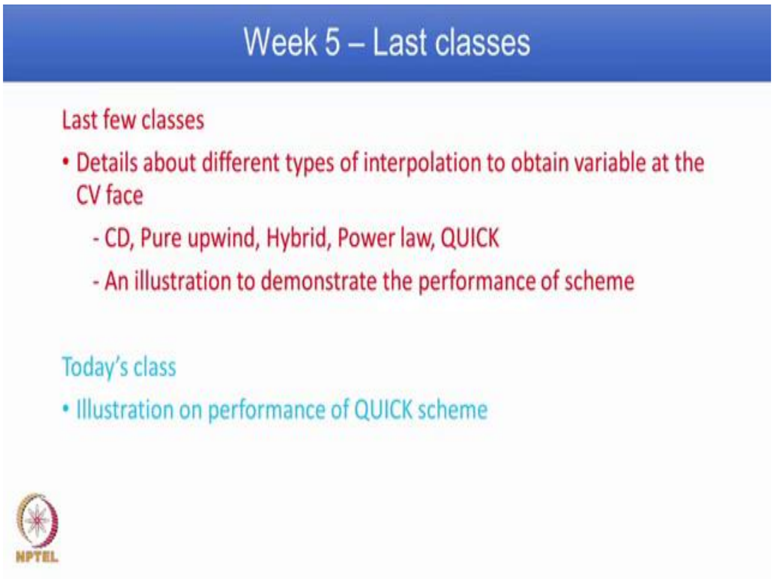
Foundation of Computational Fluid Dynamics
Dr. S. Vengadesan
Department of Applied Mechanics
Indian Institute of Technology, Madras

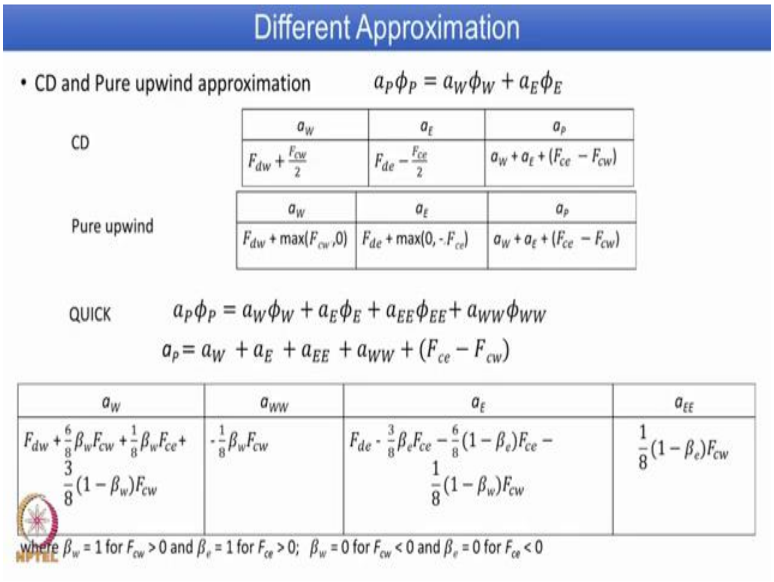
Lecture - 22

(Refer Slide Time: 00:28)



Welcome, and welcome again to this course on CFD. Today is our second module of this week. So, far we have done detailed description about different interpolation scheme available to obtain variable at face for using it in convection term. We have listed 5 different schemes central difference type that is otherwise linear interpolation scheme, pure upwind, power laws scheme, hybrid scheme and quick scheme. We also took an illustration and explained performance by central difference type scheme as well as pure upwind scheme. In today's class, we will particularly see performance by quick scheme for the same illustration.

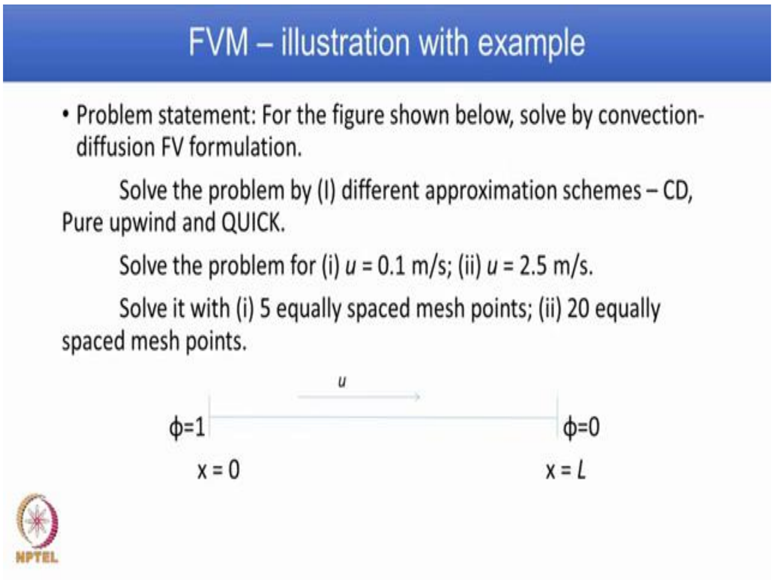
(Refer Slide Time: 01:18)



For the sake of completeness, we will list all the scheme information for CD and pure upwind approximation. The generic formula is a P phi P equal to a W phi W plus a E phi E. In the case of CD scheme, coefficients a W, a E and a P are listed as shown, and you can recognize a P that is the point of interest coefficient is the summation of neighboring coefficients that is a W plus a E plus additional term. For pure upwind, again a W, a E and a P are listed and for the sake of direction convenience and to write down one generic formula a condition is imposed a condition is included in the coefficient as shown here. For example, a W is written here as F d w plus max of F c w comma zero, which means if the flow direction is from left to right, then it is positive with respect to the coordinate direction, then max of this condition will result in considering F c w; similarly for other coefficients. And as observed in CD scheme, a P is again summation of neighboring coefficient plus additional term.

For QUICK scheme, we have similar term that is a W phi W plus a E phi E plus 1 extra node on the upstream side, which is again considering both the side possibility, the generic formula is given here. So, the final formula is a P phi P equal to a W phi W plus a E phi E plus a EE phi EE, which is east of east plus a WW plus phi WW, which is west of west. As usual, the node interest coefficient a P is related to neighboring nodes, so a P equal to a W plus a E plus a EE plus a WW and then additional term. Now, let us look at the coefficient in details; a W, a WW, a E and a EE. Again this formula has a generic condition incorporated here by including a coefficient beta. For beta equal to one, F c w is greater than zero, that mean the flow is from left to right, then it will take accordingly the coefficient values. Similarly, for beta equal to zero, or F c w less than zero, the flow is from right to left, mostly in the case of reverse flow then it will take appropriately. So, this generic formula is suitable for any direction, similarly for pure upwinding. So, this helps while writing the code, it automatically switches based on the flow velocity condition.

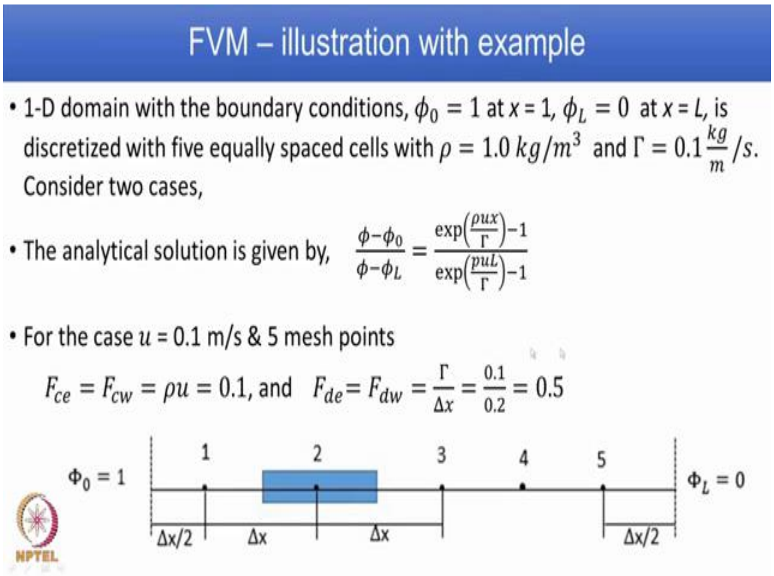
(Refer Slide Time: 04:47)



Now, let us took a problem statement once again. For the figure shown below, solve by convection-diffusion finite volume formulation. So, in this figure, there is a convection as well as diffusion for the problem, boundary condition is defined on one side that on the left side at x equal to zero, phi equal to one then boundary condition of the other side right side, at x equal L and phi equal to zero. Now, flow is from left to right, hence it is only positive, so in the generic coefficient, it will be considered accordingly. We mentioned just to consider performance by difference scheme as well as to understand influence of mesh and velocity condition, solve the problem by different approximation schemes, CD, pure upwind and QUICK. Solve the problem for very low velocity u is equal to point one meter per second and the high velocity of 2 point 5 meter per second. So, the difference in velocity will decide the weightage of convection term in the case of pure upwinding or hybrid scheme.

Similarly, solve it with 5 equally spaced mesh points or two equally spaced mesh points, so the given domain length is x is equal to 0 to x is equal to L, length- L; if L is equal to one, then if you divide by 5 equally spaced mesh points, delta x will be 0.02; if you divide by 20, then delta x will be 0.02. In this delta x term goes into the diffusion term calculation.

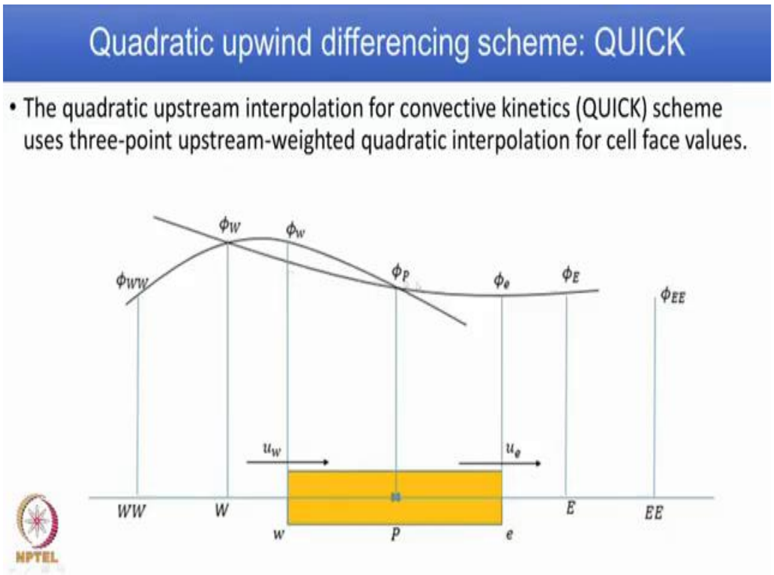
(Refer Slide Time: 06:29)



In the last slide, we explained the problem. Now, let us get some more explanation one-D domain with the boundary conditions phi at 0 equal to 1; phi at L equal to 0, when x equal to L is now discretized with 5 equally spaced cells. And for the problem define rho is taken to be 1.0 kg per meter cube and gamma taken to be 0.1 kg meter per second. Let's take two cases, and this is the domain. We discretized the domain with 5 equal nodal points, 1, 2, 3, 4, 5 and they are equally space. So, you see delta x as the distance between node 1 and 2, same way the distance between node 2 and 3 and so on, because it is equally spaced. Node one which is the node near to the left boundary, similarly node which is the node near to the right boundary, both are at the distance of delta x by 2 with respect to the respective boundary. And boundary conditions are already shown here, both on side as well as on the right side.

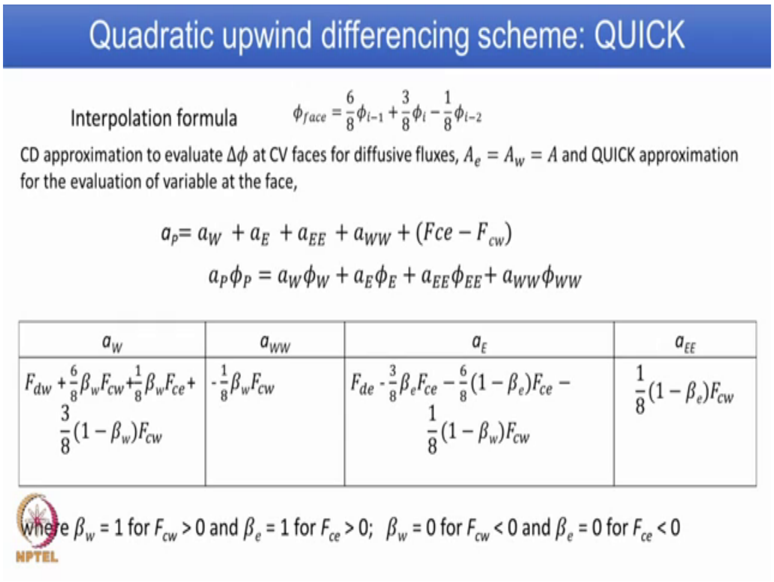
And for this problem analytical solution is already given and that is what it is displayed here. Now, we take one of the case that u equal to point one meter per second and 5 mesh points. We calculate convective flux as well as diffusive flux; convective flux equal to rho into u; rho is already defined as one kg per meter cube and u is point one meter per second, so rho u, you will get 0.1. Similarly, for diffusive flux, it is defined as gamma by delta x; gamma is 0.1, delta x is 0.2 and that is equal to 0.5. We want to evaluate convective flux as well as diffusive flux to get the idea of role of convective flux and diffusive flux. Now, this relation or weightage between convective flux and diffusive flux will inform or tell us about performance by different discretization scheme.

(Refer Slide Time: 09:08)



We know QUICK scheme illustration as shown here. This is given particularly for this illustration problem. Where the flow is only one direction that is left to right that is what you are seeing here, both u w as well as u e are shown from left to right. So, for the face w, phi w is your interest phi small case letter w is your interest, then you have a quadratic fit running through 3 nodes. Say immediate downstream node is phi at P, the one node on the left is phi at W capital letter, then one node upstream is phi WW that is the quadratic fit. Similarly, for face u e, then you consider one node immediately downstream is phi at capital letter E, the one node on the other side phi at P, then one more node immediately upstream phi at W that is the graphical representation of quadratic fit.

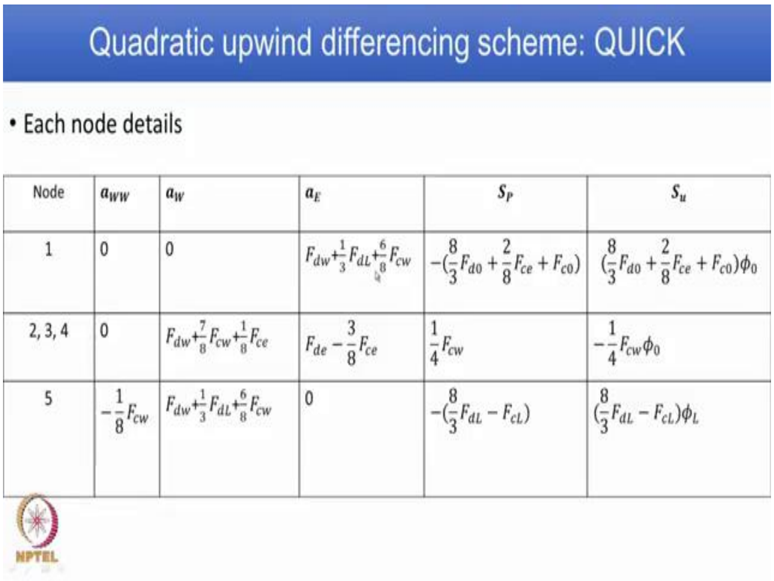
(Refer Slide Time: 10:13)



Now, we write down the generic formula phi at any face is as shown here. As I already mentioned, in this formula, node the letter subscript i represent immediate downstream, then i minus 1, and i minus 2 are upstream. The CD approximation to evaluate delta phi at CV faces for diffusive flux is used, because in this problem, we have convection term as well as diffusion term, and diffusion term, we mention we will use only central difference type approximation. We already did that in detail.

Now, again for this problem, area is the same, so A e equal to A w, now we are using QUICK scheme approximation for the evaluation of variable at the face. Now, after doing that operation and putting in both the final discretize form then you will get in the regular form, final discretized equation as shown here a P equal to a W plus a E plus a EE plus a WW plus remaining term. And for a P phi P is related to neighboring coefficients as shown here. When we already listed this a W, a WW, a E, a EE; in this problem, the flow is only form left to right, so accordingly the coefficients are taken only suiting to the problem. And beta is defined as we did before.

(Refer Slide Time: 11:49)



Quadratic upwind differencing scheme: QUICK

- Each node details

Node	a_{WW}	a_W	a_E	S_P	S_u
1	0	0	$F_{dw}+\frac{1}{3}F_{dL}+\frac{6}{8}F_{cw}$	$-(\frac{8}{3}F_{d0}+\frac{2}{8}F_{ce}+F_{c0})$	$(\frac{8}{3}F_{d0}+\frac{2}{8}F_{ce}+F_{c0})\phi_0$
2, 3, 4	0	$F_{dw}+\frac{7}{8}F_{cw}+\frac{1}{8}F_{ce}$	$F_{de}-\frac{3}{8}F_{ce}$	$\frac{1}{4}F_{cw}$	$-\frac{1}{4}F_{cw}\phi_0$
5	$-\frac{1}{8}F_{cw}$	$F_{dw}+\frac{1}{3}F_{dL}+\frac{6}{8}F_{cw}$	0	$-(\frac{8}{3}F_{dL}-F_{cL})$	$(\frac{8}{3}F_{dL}-F_{cL})\phi_L$

Now, we apply this generic formula for each node separately. As we mentioned while describing the problem figure, node one and node 5 are near boundary nodes; internal nodes are 2, 3, 4, so 2, 3, 4 nodes we have the same expression and that is what is shown here in this table, 2, 3, 4 then evaluate then you get this. For 1 and 5, there are boundary nodes, and we have already explained the left side of the boundary node 1, and right side of the boundary node 5, actually the boundary condition then accordingly you write down the expression coefficient expression. Now, we have to substitute values that is given in the problem statement in terms of rho gamma, velocity as well as number of mesh point that you have consider. So, if you substitute those values in this flux calculation properly, for example, F d w means diffusion flux at west face; similarly diffusion flux at L and so on.

(Refer Slide Time: 13:04)

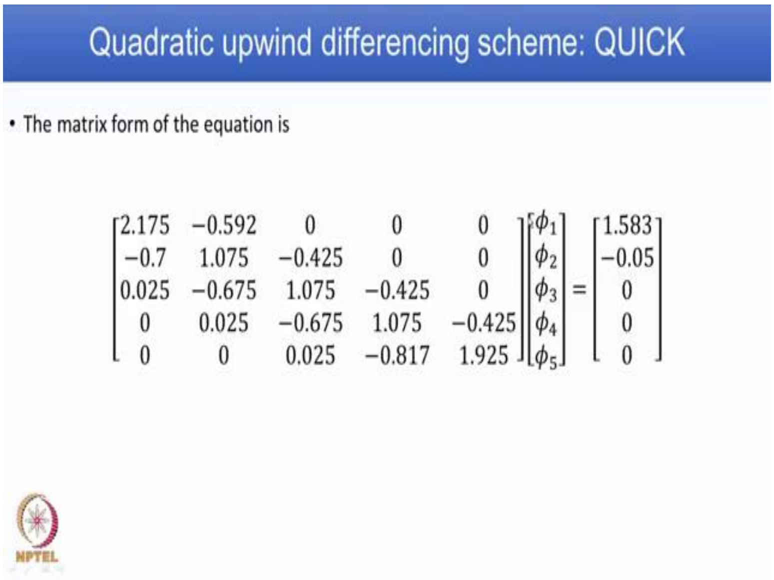
Quadratic upwind differencing scheme: QUICK

- For the problem with $u = 0.2$ m/s. $F_{ce} = F_{cw} = 0.2, F_{de} = F_{dw} = 0.5$.

Node	a_{WW}	a_W	a_E	S_P	S_u	a_p
1	0	0	0.592	-1.583	$1.583\phi_0$	2.175
2	0.0	0.7	0.425	0.05	$-0.05\phi_0$	1.075
3, 4	-0.025	0.675	0.425	0	0	1.075
5	-0.025	0.817	0	-1.133	$1.133\phi_L$	1.925

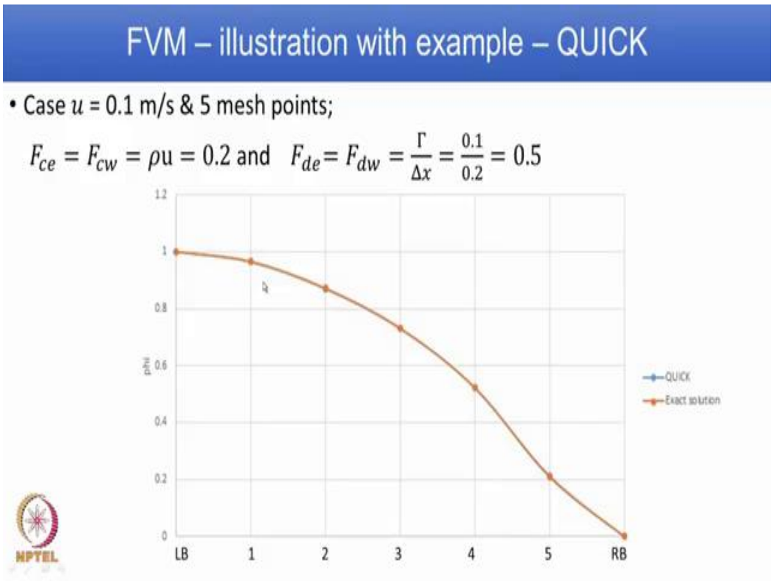
You substitute all those values then for this condition u is equal to 0.2 meters per second then evaluate and you get this coefficients actually calculated. So, for node 1 as well as node 5, we have a source term coming from the boundary condition. And for node 2 which is next to the near boundary and that will have again a west of west, because in this problem, the flow is from left to right, when you write down for node 2, then QUICK formula will have a P, a W and a WW, a WW, which is west of west. And in this problem, it happens to be the boundary condition face so that will also appear as the source term, and that is what is shown here as S P and S u. Node 3 and 4 is actually deep inside, and they do not have the boundary condition incorporation. So, S p and S u will take the value of 0. So, this numbers are obtained after substituting problem statement value that is u is equal to 0.2 meter per second, delta x based on the condition that 5 mesh points are used; gamma is already given, rho is given, then if you substitute in the generic formula and take appropriately the boundary condition then you are able to evaluate and get this values.

(Refer Slide Time: 14:43)



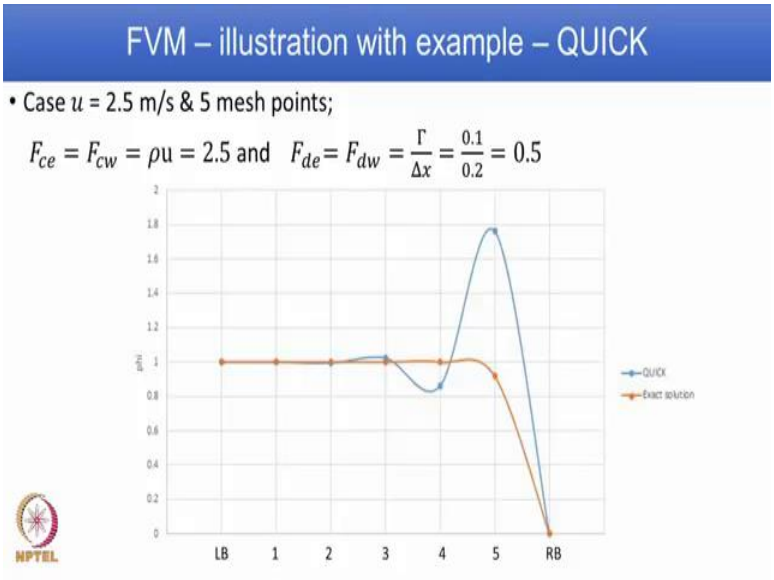
Now, this needs to be put in the form of matrix so that is what you will see here. So, in the form of a matrix it is shown. There is one coefficient matrix then there is one unknown column vector, and that is equal to another known column vector. So, in this phi one to phi 5 are nodal values to be determined and corresponding coefficients are written in this Richardson coefficient matrix. As I explained before if you look at this matrix particularly, we have one diagonal element, so in this case, it is 2.175, 1.075 5, 1.075, 1.075 and 1.925 these are all element along the diagonal. Then you have immediately super diagonal, then there is one immediately sub diagonal so that is the tridiagonal matrix structure. But for this problem, when you use QUICK scheme, you get additional element, so the tridiagonality matrix structure is actually disturbed, this also we explained before. Now, we are seeing it actually with the problem and corresponding numerical values. There is a procedure to invert this matrix, we will reserve that matrix inversion procedure for next class. So, once you do this matrix inversion, you are able to get answer or unknown variables phi 1 to phi 5.

(Refer Slide Time: 16:15)



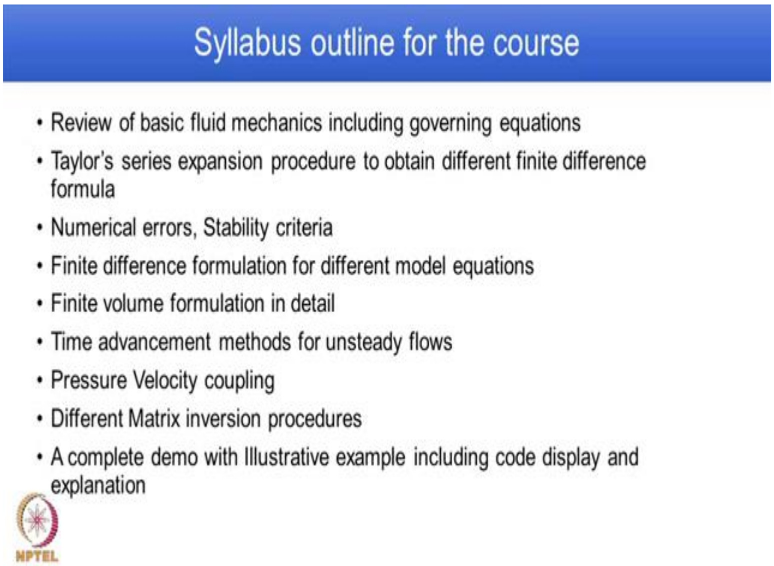
Now, we have already mentioned there is a analytical solution available and you can make a comparison. Now, this is comparison on the performance for the same problem, but using a different value of 0.1 meter per second. The previous slide I showed values corresponding to 0.2 meter per second and this is slightly lesser point one meter per second, but everything else same that is 5 mesh points, rho and gamma. And you see here performance, QUICK scheme plot and exact analytical solution plot they are shown, and in this case, one exactly matches with the other and that is why you are seeing only one color. And you can cross check the left boundary, it is already given as one, in the right boundary it is 0, and there are 5 nodal points 1, 2, 3, 4, 5. So, you are able to compare and you see performance because it is so low velocity QUICK scheme performs equally well.

(Refer Slide Time: 17:23)



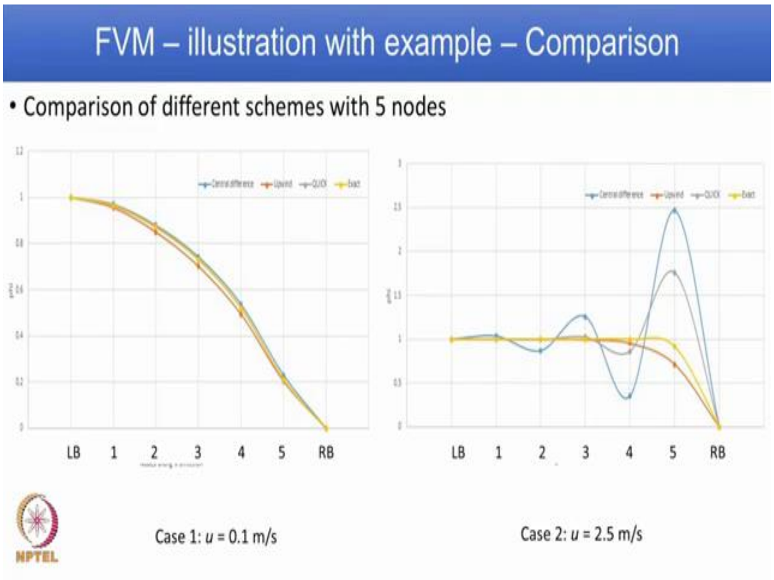
In the next plot, you see performance of the quick scheme, if you increase the velocity from point one meter per second to two point 5 meter per second, number of mesh points still remains as 5, then you see the performance of the quick scheme. So, this color, the red color one is exact solution, whereas, the quick scheme start showing from node 3, node 4, node 5 and then there is a right boundary. This is because you are having a 5 mesh points and delta x is 0.2 and you compare convection is 2.5, and diffusion is 0.5, and performance of quick scheme is poor. Now, we will see in the next slide same flow condition that is 2.5 meter per second, instead of 5 mesh point for the same domain, increase at 20 mesh point.

(Refer Slide Time: 18:21)



And again rho u value remains, because it is function of velocity and density is two point 5. Whereas diffusion is changed gamma by delta x from 0.1 by 0.05 is equal to 2.0. So, diffusion is now 2.0, which is almost of the same order as a convection, 2.0 is the diffusion flux and 2.5 is the convection flux. Now, you see the performance of the quick scheme for the same problem the results obtained are better, and it is very close to the solution obtained by quick scheme. So, this plot tells you how important it is though the scheme is correct, if you increase the mesh size then you are getting a better solution or correct solution. So, it is necessary to check all the parameters before coming to the decision whether scheme is good or flow conditions is not properly imposed. We have seen a similar one, last class when we demonstrated CD scheme, similarly when we use 5 mesh points for higher velocity, the CD scheme was showing a vigil that is oscillation and when we changed from 5 mesh points to twenty mesh points, solution improved.

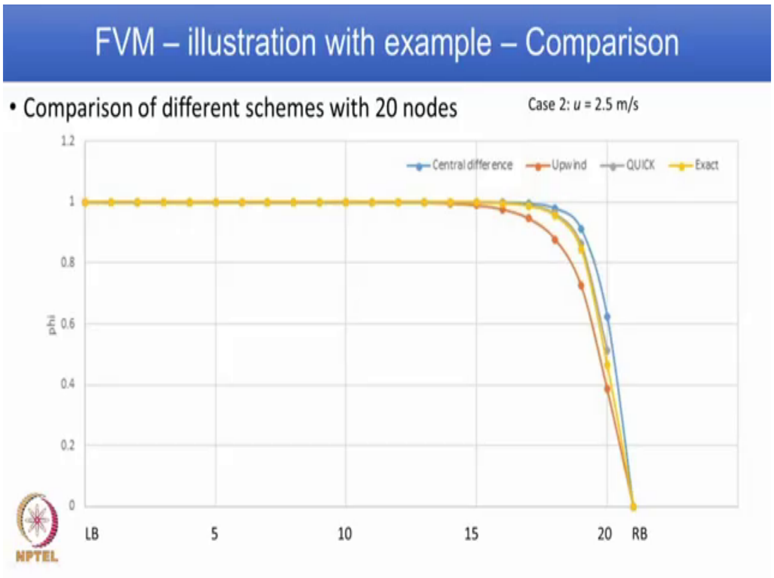
(Refer Slide Time: 19:44)



Now, in this slide, I am going to make comparison of all the schemes. First one is comparison for the flow condition, u is equal to point one meter per second, this flow is very low velocity, hence the scheme is not much influence. Even if you use 5 node points, you see here performance by all the schemes almost matches with the exact solution and this not much difference. Now, if I increase the velocity from point one meter per second to two point 5 meter per second, you can see the comparison for the same 5 nodes, you can see here exact solution is marked by this color. Then upwind scheme is marked by a red color and you have a QUICK scheme and then this is QUICK scheme and then we have a SIMPLE difference scheme. We can make a interpretation with this plot itself, upwind scheme is almost matching with exact solution and then central differences scheme showing a very big variation or very big vigil, whereas, QUICK scheme when you use a QUICK scheme though the performance is not matching, you are able to see the vigil get reduced. Right from central difference scheme it gets reduced this much. Similarly, on the other side, it gets reduced from to this side to this level.

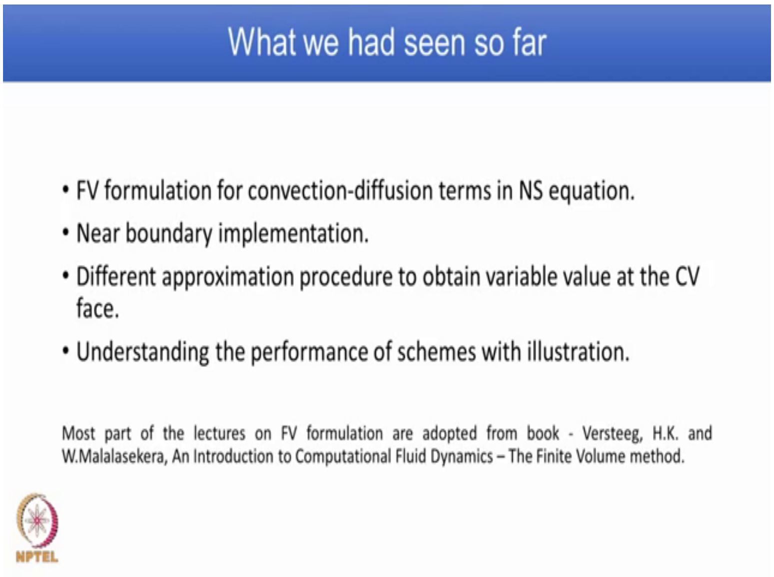
And you can understand QUICK scheme is kind of combination of CD as well as pure upwinding that we already mentioned when we explained the QUICK scheme. It has two nodes on either side plus one node on the upstream side. So, two nodes on the either side, behaves like a CD and one extra node on the upstream side behaves like a pure upwinding. So, it has a influence of pure upwinding as well as CD scheme that is information is very well captured in this comparison illustration, the performance by QUICK scheme is better, though it is not good, but the vigil get reduced and it is getting close to the solution. And this much reduction, you can interpreted, because of the pure upwinding type introduced to the QUICK scheme.

(Refer Slide Time: 22:12)



And if you increase the mesh size for the same problem from 5 to twenty, then all the schemes behaves very well. Then all are able to compare results are exact solution.

(Refer Slide Time: 22:28)



So, in this lecture, last the few classes, we have seen in detail finite volume formulation for convection and diffusion term in Navier-Stokes equation. We also learned how to do near boundary implementation. We learned there is a need to do different approximation to get the variable value on the face. We learn 5 difference scheme, we also learn the performance by taking one illustration. And by increasing the mesh size or by increasing the flow condition performance differs. Most part of the lectures on this finite volume formulation are adopted from book The Finite Volume Method authored by Versteeg and Malalasekera.

So, next class, we are going to see another interesting topic until then have fun.

Thank you.