Computational Fluid Dynamics

Prof. Sreenivas Jayanti

Department of Chemical Engineering

Indian Institute of Technology, Madras

Module No. # 07

Dealing with complexity of geometry of the flow domain

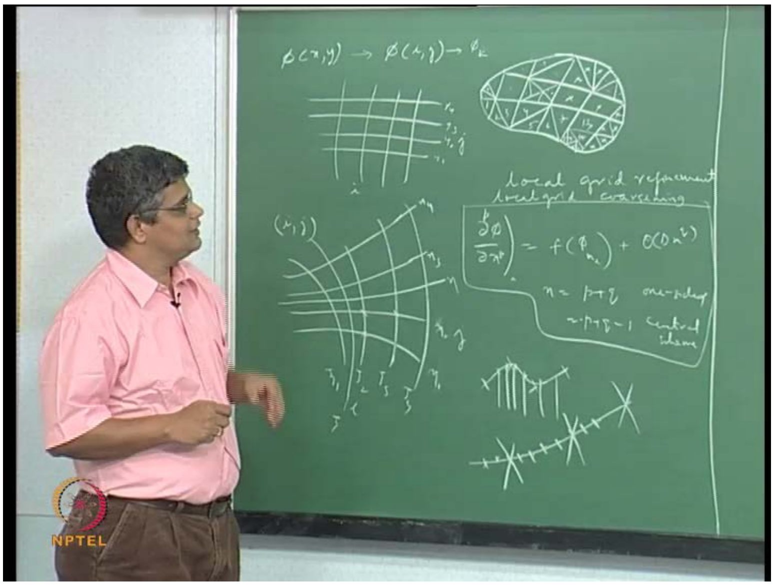
Lecture No. # 43

Topic

Finite volume method for the general case

Now, let us look at the finite volume method in the more general context, where we take the conservation equation in the fullest form, and then try to convert it into, write it down in a conservation form, and then convert it into the flux formulation, and then look at how to evaluate the fluxes, and how to evaluate the areas and surfaces, so that using, by applying this conservation equation on a single control volume, we will be able to come up with the the conversion of that in into an algebraic equation for the value of the variable in that particular cell. So, that is that is what the finite volume method does. It does not have too much of an effect on the rest of the process of the cfd solution. It defines, it takes a different approach to converting the partial differential equation into an algebraic equation; so the details of this for the general case, where we do it in systematic way, so that there is no confusion arising, when we evaluated the surfaces and the fluxes.

(Refer Slide Time: 01:36)



So, we are looking at a case, where we have some domain, closed domain without any leaks like this, and this is made up, this is broken up into some sort of tiles. And the idea is that, one can have a mixed set of tiles like this, so that the surface here can be faithfully represented. So, the each of this is a control volume, and by packing enough of this, we can make sure that the surface is properly represented. So, the overall bounding surface of this forms one of the sides of some of these control elements, and if in two dimensions, if you want to use some triangular kind of thing close to the surfaces, you can be sure to put to represent even a fairly complicated surface properly by selecting it in this way.

So, in the in the general case, we have the flow domain decomposed in two dimensions into set of quadrilateral and triangular elements. And when I say quadrilateral, I do not necessarily mean rectangular, you can have distorted quadrilateral; and triangular it would be ideal to have it as an equilateral triangle, but it can also be distorted to triangle. So, so it is made up of this, in such a way that when you put all the tiles together, when you put all the tails together, you take up the entire area. In the case of three dimensions, we can use a combination of tetrahedrals, and then rectangular blocks, kind of things to make up the overall volume.

At this introductory stage, we will not really look at the three-dimensional aspect, but we can take a two dimensional thing, and we can see by making the triangle smaller, and then larger as necessary to represent the curvature. We can have a fairly faithful reproduction of the entire cross section in these two dimensions, in which you have the, which constitutes the flow domain, the computational domain of interest, and within this we want to find the variation of phi.

We want to for example, find the variation of phi as the function of x and y throughout this; and as usual in a cfd, we do this by writing, by evaluating phi at several points i, j not at every point x and y, but at several points i, j. And this notation of i, j here is appropriate for a structured grid, just as an x, y grid associated with cartesian mesh. You go a certain distance in the x direction, you go a certain distance in the y direction, you get to that. So, this i, j notation we have used is for a a structured mesh, you go through so many mesh points in the in the x-axis, and so many mesh points around the y-axis direction to get that particular thing.

So, this notation here is not very relevant for a for a unstructured mesh that we normally use in finite volume method, because there is no specific i direction and j direction in this. So, associated if you say that this is the overall flow domain, two-dimensional flow domain, which is decomposed into this, these individual tiles, cells. Then for each of these cells, we have a particular point; the centroid at which the value is determined. So, these are the points, and we say that this value times this particular area is the total amount of the particular variable in terms of the intrinsic properties. So, the this is per unit area per unit mass type of thing. So, that is contained in this. And one can see that, there is no i, j, k orientation in this; you cannot say that this point is along i, first point, second point, third point like this, and this is not on the on any intersection like these things.

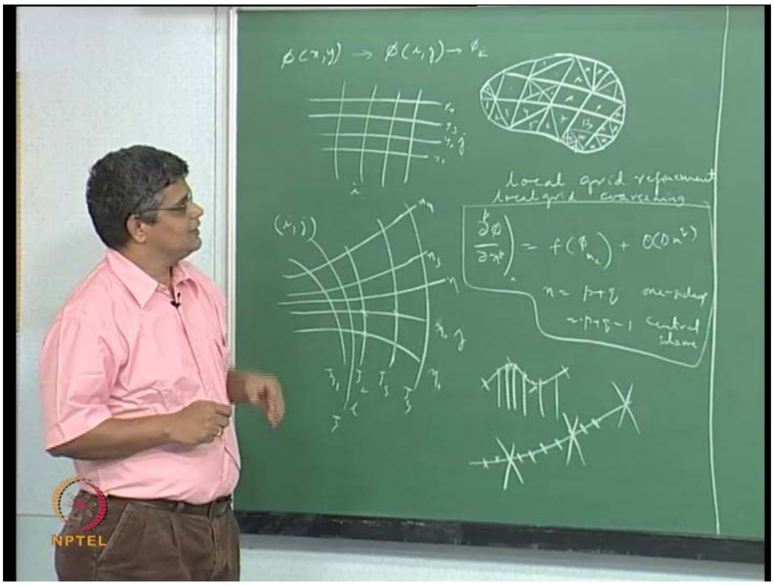
So, if you have like this or like this, these are the family of coordinate lines x and y, and these are the general coordinate lines for example, psi 1, eta 1, eta 2, eta 3 and eta 4; psi 1, psi 2, psi 3, psi 4, this is what we have for the general body fitted type of grid, and this is what we have for the cartesian mesh y 1, y 2, y 3, y 4 like this. And then one can associate an i and j, so for example, an an i here, and the j here, will define this particular point, and i plus 1 means, j is this point, and i, j minus 1 is this point like this. So, one can define an i, j type of identification, for each point on a structured mesh. So, this kind of i, j notation is no longer valid in an unstructured mesh, which is which is a mixture of for example, quadrilaterals and triangles.

And even when you make it entirely of triangles; in the general case, it is not possible to come up with an i, j kind of notations. So, we do not when we are talking about the discretization of this, we do not use this i, j type of thing may be we use just k, a single thing, which identifies the particular element number. So, when we do this, making up of these tiles, we need to have some numbering scheme associated with this. So, we may have 1st, 2nd, 3rd, 4th, 5th, 6th, 7th, 8th, 9th, 10th, 11th, 12th, 13th and so on like this. And the idea is that we are evaluating in the process, we are evaluating only the value of phi at the centre of each of this, each tile have a will have a single value of of phi associated with its centroid or some other convenient position; and so that is the value that we have.

So, when we talk about the value of phi here, the known values; we know the value of phi here, here, here, and here, here, here, here, here. So, there is no specific pattern has to where this things are available. So, and that is why it is more difficult to come up with specification of derivatives, which are required for the flux terms, and the evaluation of the appending fluxes that are associated with the with with the advection term. So, in such a case, that is why it is very difficult to say, we know that a scheme a derivation derivative of of p eth derivative expressed in the form of phi neighboring i's, at i to the order of delta x q will require a number of points as p plus q, in the case of one sided; and p plus q minus 1, in the case of central scheme; all this is valid for a structure mesh.

And if you were to say that I want to evaluate the flux here, at this point you see how much is coming into this; and I need to evaluate that with with a fourth order accuracy. Then if you are using a central scheme, we need second derivative, so we need 5 points. So, we may want to have this point, this point, this point, this point, this point. So in this particular case, it has come out that they are more or less on the same line, but if you looking at here or this point 1, 2, 3, 4, 5, they not on the same same lines. So, that is why it is difficult to come up with higher order accurate schemes, for the evaluation of the fluxes in in a finite volume mixed type of mesh like this, because there is no identification of i and j; and this is the structure that is last, then we go to an structured mesh; and that is why we cannot make use of these these kind of approximations of arbitrary or decide level of accuracy, that is very readily possible in a structure mesh; and that is why most in the finite value method, in most of them, the discretization of second order accuracy is considered to be good enough.

(Refer Slide Time: 01:36)



And if you want more accuracy, what you do in a finite volume method is to reduce the number of cells, by doing this, you are creating two cells, so the evaluation is now at this point, this point. Now if you want more accuracy, you can you can make it like this, and you can make it this way, and then you can make it even less. So, you can arbitarily locally refine the grid, so that you can get a more number of points in in a particular zone of interest. So, this kind of is by which local refinement is possible in in a finite volume method is not there in the case of structured mesh.

Here if you want to have the accuracy level increased by reduce the grid grid size, you have to do like this, you have to introduce one more psi line here, and you have to introduce one more eta line here, and this introduction here not only introduces cells for example, this has reduce the cells as into 4, but it is also done that here, it is also done that here, and it will be carried all the way to the boundaries on this side and this side, and this side, and this side. So, you have not doing any local refinement, you are doing refinement, which is stretching far beyond the boundaries that are far beyond the zone of interest, and that becomes a point of weakness in the case of structured meshes.

So, from that point of view, when you look at it from the overall point of view of accuracy of the discretization solution, structured meshes have the advantage of of these kind of higher order accurate schemes, a approximations for derivatives being implemented, they can be implemented in in structured meshes. Whereas, in the case of unstructured mesh we do not have the luxury of coming up with higher order schemes, but you have the luxury of locally refining the grid by doing these kind of a a breakups. It is not as trivial as its sounds hear, they has to be it has to be done in a very systematic way, but algorithms are available by which you can do this in in a systematic way, and you can easily do a local grid refinement .

So, and it is truly local in the sense, you can concentrate on a particular area, and you can make it refined; without having to worry about the same kind of refinement propagating into the other sides. And you can also do local grid coarsening; in a typical cfd solution, we are going to get estimates of the solution, because we are using iterative methods. So, that means that before we get the final solution, we already have some idea of what is going to be the rough shape of the solution. So, in such a case, we may want to do, we may want to adapt the grid that we have used, because the grid in a way defines the order of, the level of accuracy of the solution.

So, where the gradients are large, you may want to reduce the grid, reduce the delta x type of thing, here we have to reduce the size of the grid. And where the gradients are very less, where there is no variation for example, if you have if you are expecting variation like this, you need to have a small delta x. So, if you have computed a value, which is fluctuating like this, then in order to truly seek this this variation, you may want to put more number of points here.

And if you have got a variation almost like this, and if you have a gird which is like this, then looking at the solution and says that there is hardly any variation, why not I use a large grid, because large grid will reduce the computational time (( )), and the small grid will increase accuracy. So, this always the struggle to to to make the two counter requirements that that you have you would like to have as small a grid, so that your computation is faster; at the same time you want to have as finer grid as possible, so that you accuracy is high. So, that kind of adapting the the grid locally to take advantage of coarsening possibility where the variation is not very rapid or the need for accuracy in a case of rapidly varying quantities by doing grid refinement. So, this these kind of things are readily possible in a truly local sense in the case of finite value method.

So, this this is one of the advantageous of the finite volume method and the grid adaptation is one big that is considered as as as probably the reason going for finite volume method; and obviously, that the the possibility of dealing with a complex geometry like this, without having to try to get a body fitted gird, which which fits these things and all that. So, there are specific reasons for this. So essentially, there are possibilities, but you can also see some difficulties that are coming here; especially from the point of view of discretization that we would have to consider.

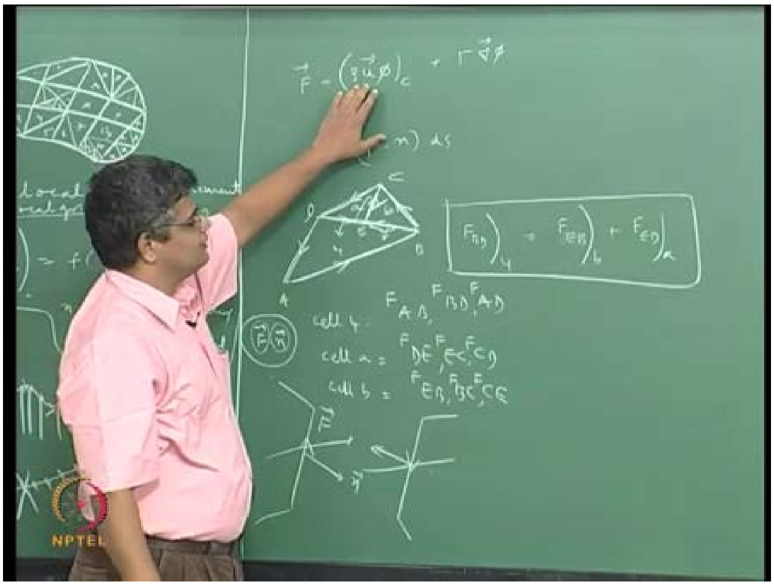
When we talk about the fluxes, when we talk about the finite volume method, we are talking in terms of volumetric source terms and surface based flux terms. So, we are distinguishing primarily between volumetric terms, which can be integrated over the entire control volume; and fluxes which have to be evaluated at the the at the surfaces at surface of the closed domain constituting that particular control volume. And the control volume need not be only rectangular or only quadrilateral or only triangular, it can a combination of those things, as we have seen here.

And what we also see here is that in this kind of meshing and with grid refinement and all that, it is not necessary that one face of this triangle must be shared by one rectangle or another triangle like this; one can see between their control volume 6 and 7 the two triangles, this face is being shared by both of them. But in this case or in this case here, this particular phase is being shared by 2 other things, and this particular phase is being shared by two other elements.

So, that is something that that has to be evaluated properly, because we always want to have when evaluate the fluxes, we want to have flux balance. The flux that is leaving a control volume through one of the phases must go into the adjacent phases to the same by the same amount; if a flux of 100 is leaving the control volume of of one particular phase is the control volume, then this 100 must go into the adjacent control volume through that particular phase or adjacent control volumes through that particular phase.

And this is very important, because for in our as we have seen in in the simple example, we evaluate for each control volume, for each control surface we evaluate the fluxes, and then we make a balance out of it, based on that we we convert the partial equation into algebraic equation. But so, when we deal with each control volume separately, and when we for example, when after discretizing this, we go to the discretization of this element here; this element has three surfaces, and through each of them, there is a flux. And so, part of the flux, which is coming through this phase is coming from from this is leaving this particular phase of the control volume.

(Refer Slide Time: 21:51)



So, now we have an element 4 here, let us say that this is a and b; and then we have we have 2 triangular control volumes, which are adjacent to this, and if you now put a A, B, C, D like this and E. So, this surfaces for cell 4 are AB, BD and AD; and for cell a, the surfaces are DE, EC and CD; and cell b it is EB, BC and CE. So, we must when we evaluate the fluxes for phase 4, so you have a flux associated with AB, flux associated with B D and flux associated with AD. And similarly, we have the flux components associated with each of this these things; and in the evaluation, there must be a condition, which says that F BD for cell 4 that is this one, the total surface, flux, which is leaving through this surface is equal to the flux EB for control volume b plus the flux ED for control volume a. Only then, one can say that the flux that is leaving, this control volume is going into the adjacent control volumes; if this is not satisfied, there is a mistake, there is a imbalance between the flux leaving this and flux coming into this.

So, when we evaluate the fluxes for each of this these things, we have to make sure that this kind of condition satisfied. And I am also being very loose in making this definition, because we are evaluating the flux through each of these, and in the process we have to look at the flux direction, we have to see that we are taking when you say that we have used F for the flux dotted with n. So, we will see when we put a proper notation, f is a flux is the for example, this is the gradient, and n is the normal vector. So, we have to evaluate the dot product here, so that means that we have to have an idea what is the outward normal vector for each of the phases. So, we we have to see that this particular phase here, when we consider 4 is the outward normal vector in this direction. When you consider element a here, the outward normal vector is in this direction; again for b, it is in this direction. So, we have to make sure that the evaluation of each of these fluxes is done with the appropriate orientation of the outward norm vectors.

So, and one way of ensuring this is to for each cell, when we define the phases like this, we always take the counter clock wise direction. So, we talk about F AB, F BD and F DA; again for this particular cell here, we go in the counter clock wise direction, so we say BCE and then back to B. So, this goes in this direction, and here it goes in this direction, and here it goes in this direction. Now for cell CDE, if you again apply the counter clockwise direction, it this goes like this, this comes here, and this comes here. So, when you are when you are looking at cell 4, you are traversing this whole length in the counter clock direction that is in this direction like this, from B to E and D to E E to D, but for this cell, we have traversing this in the opposite direction, so this part will get cancelled with this; and again for this, you are traversing this in the opposite direction, so this part will get traversed in this.

So, by consistently using counter clockwise direction to define the bounding surfaces of each element, we can make sure that this sort of the outward normal vector for each phase is identified. So, if you have a phase like this, and this is a control volume, so that we can see the control volume is to the left of this, then this is the outward normal vector of this particular surface. So, the outward normal vector rise to the right of… So, if if the control volume is to the left, then this goes to the right and we are going in the counter clockwise direction. So, if the same surface, the control volume is in this direction to the right here, then if you go in this direction, we are going in the clockwise direction. So, by adopting it in by going through this control volume in the counter clockwise direction, the same distance or the same phase is is being traversed in the downward direction; that means that the outward normal vector is again in this direction.

So, it is it is moving, it is if you are traversing in the direction of the counter clockwise direction, the surface normal vector is to the right, to a right and the control volume is to the left. And if the flux vector is aligned in the outward normal direction, then it is flux leaving this; and if the flux vector is aligned in the opposite direction, then it is flux coming into this. So, here if you have flux direction here like this, then this flux dotted with this, one can see that it is leaving the surface; the same flux vector acting on this will be a flux, which is coming inside. So, in that direction, in that sense, the flux dotted with that proper surface normal direction will make sure that will automatically consider whether it is flux that is coming in or going out, and we have to make sure that the evaluation of the flux is done consistently, it is a vector quantity; and the evaluation of the outward outward normal vector associated with that phase is also done properly.

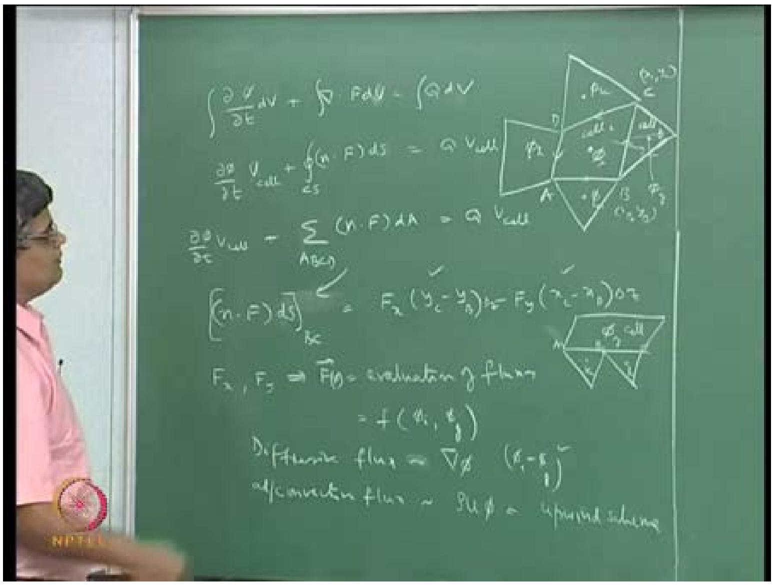
So, it is important to retain this directionality of the defining of the definition of the phase in order to maintain this, the flux is always as per the for example, if you say the flux has some rho u phi, which is the convective component, and the only vector is the rho here. So, this follows the convective flux follows the direction of the of the velocity, and the diffusing flux is typically like some gamma gradient of phi. So, this follows the gradient direction. So, the the direction of the flux is fixed by the velocity in the case of convection, and the gradient of phi in the case of diffusive flux, and this has nothing to do with the area and all these things; it is nothing do with the geometric of the cell. But the normal component of this is has everything to do, it has only thing to do with with the orientation of the of the surface, and by maintaining directionality here in the counter clockwise direction consistently, one can also use counter clock clockwise direction consistently.

But for each element, we have to make sure that the element is defined in the in the clockwise direction or anticlockwise direction, and the evaluation of fluxes must be done in the way that is consistent with the with the evaluation evaluation of the individual components; velocity defining the direction of the convective flux; and the gradient of phi determine this. So, by evaluating this, then we can make sure that this condition is implicitly satisfied, plus one more condition here that the evaluation of fluxes must be the same for this particular phase, whether we are considering it has an outward flux or inward flux. So, that is the evaluation of fluxes should depend on the phi values in a consistent way that when you are considering for example, this surface here, this control volume here, you are trying to evaluate the flux coming through this.

Now, that flux obviously, depends on the value of phi at this point. Now, when you consider the flux, which is coming through this point, for this cell here; then if you evaluate it using the value here, then there may be some inconsistency in in the evaluation of flux here, in terms of Fourier phi and gradient phi, when you consider this phase as being belonging to this, and as this phase being belonging to this. So, the definition of the evaluation of the flux must also be consistent that the quantity of F that is being attributed for example, to this phase is the same, whether you are considering this this cell or this cell.

And similarly what should change is obviously, the direction of the outward normal vector for each of this; only then we can make sure that the consistency in the flux and overall conservation of fluxes satisfied. So, this is one important element in the overall discretization of the control volume; if this is not satisfied, then you will you'll have the great difficulties. Now, let us look at the evaluation of the fluxes, we have looked at how that area has to be evaluated, and in order to preserve the flux, it is important that the flux is also determined in such a way that when we consider a particular phase being a part of belonging to one particular control volume, and an adjacent control volume, the actual value or the flux must be the same, must be evaluated to the same. So, how is it possible?

(Refer Slide Time: 34:23)



Let us take the case of two dimensions, and let say that we have dou phi by dou t, this del dot F, the flux equal to some Q as a source term; and we are integrating with respect to volume, and we are putting here in the flux all the terms, which will appear on the surfaces, so that we can write this for a particular cell as dou phi by dou t. Now, since we are concentrating on a two-dimensions, we will call it integration over the area, and this value will become let us say that area of this cell is A here plus n dot F times d l over the control perimeter, over the closed surface; it is put volume here, and we understand implicitly what we mean by.

We understand implicitly that in the third dimension, the length is unit value, control surface equal to 2 times volume of the cell. So, this we evaluate as dou phi by dou t, volume of the cell plus sum over for example, the size ABCD of n dot F dA equal to Q times volume of the cell; and our particular this may be our cell here with a value of phi i given there, with the some other cell adjacent to this some other cell here. So, this is all part of part of the interior part of the flow domain; and we are looking at one particular cell which is denoted by ABCD here.

And so we can take as usual this sort of counter clockwise direction; and we can expand this as n dot F times… now n here times if you consider, let us say surface BC the line BC here. So, the integral of this over BC plus integral over AB plus integral over CD plus DA like that. So, this can be written as if you consider the flux to have x and y components, and this one also to have a surface vector, which is oriented like this, and let us say the flux vector is oriented like this. We have F x and F x dotted with S x, and that can be written as F x times y C minus y B minus F y times x C minus x B. So, this this is the n product is a scalar, and for the case of 2D, where B here has (x B, y B) as the coordinates and this as (x C, y C) as coordinates. Then we can say that this value can be expressed in terms purely of the coordinate points. And this is where the evaluation of the area through which flux is passing based only on the coordinate points will be useful, because when we evaluate the flux for this particular control volume, and this will be in the negative direction.

So, and at that point it will become for example, this become y B minus y C and this will become x B minus x C, so that whatever that is treated as positive for this will be treated as negative. So, the the different directional sensitivity of the area vector is coming in this definition, where by the overall area through which flux is passing, each of this is multiplied by delta z, which is unity and which defines that surface. So, in that sense this satisfies the constant that the area evaluation that is delta CB delta y C B times delta z is done in such a way that for counter clockwise directions, it is one value; and for clockwise direction, it will be ah it will be the opposite with equal magnitude.

So, and we can similarly write for each phase with CD and DA and AB evaluated in in this way, you can take the end the vertices of the points, which define this control volume, and then we can come up with the area evaluation like this . So, in that way, we still have to worry about what is F x, and what is F y? Essentially, the evaluation of fluxes is the remaining part; and we know that the fluxes also have to be evaluated in such a way that when we talk about this particular phase, whether it belongs to this control volume or this control volume, the amount of flux calculated should be the same.

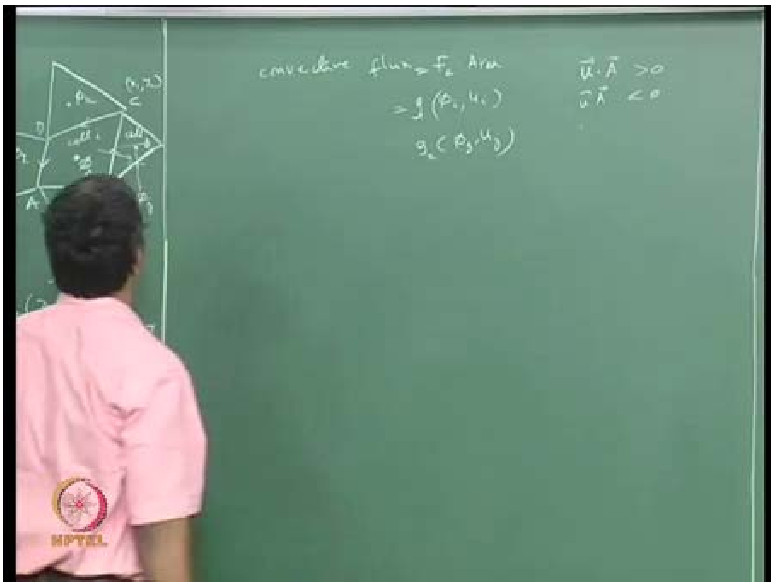
So, if that is, if that condition were to be satisfied, we have a phi i value associated with this, and we have phi j value associated with this particular cell; if you call it as cell i and cell j, with BC as the common phase through which flux is passing trough in exchanging between two things. Then this evaluation of fluxes, which is typically a function of phi value, do you have the diffusive flux, which depends on the gradient, and the convective flux, which depends on the velocity and also the phi value of that particular scalar. So, in that sense, this is a function of phi; and the evaluation must be such that it should be using the same values of phi to evaluate the flux. So, one possibility is that you can write this as a function of phi i and phi j, where phi i and phi j are the values of the variables across which this particular phase is common.

So, when we evaluate this, the flux through this, then we can we make use of this value, and for example, phi k and phi l and phi m. So, whenever we look at evolutional flux trough this phase, then we make only these values and we do not bring in this. And in such a way, when we make evaluate the flux trough this phase, which is common to these two; then we can make use of this. Now the same argument can be extended to case where for example, this particular phase is being shared by two cells j, k and l. Then in in this case, the evaluation of the flux through this particular phase for cell j must be such that this component is evaluated separately, and this component is evaluated separately with the corresponding points let us say ABC as per the corresponding areas coming into picture here.

And when you compute the flux through these two phases, these two cells through this phase, we make use of j and k here; and for this boundary, we make here of these two phase, and then add them together to get the overall flux through this phase. Again when we do that, we are making sure that the amount of flux, which is evaluated, is consistent for those control volumes, which have that particular part as one of the boundaries. So, if you do this, and if you make sure that the area evaluation is based only on the vertices, which are common vertices, then the overall flux conservation equation can be readily established.

So here, we are now we are now decided that the flux should be evaluated for this thing, through the two common through the cell values of phi at these two things. So, if again in flux, we have diffusive flux; and when we talk about diffusive flux, we are looking at evaluation of gradient, gradient of phi, and this gradient can be definitely estimated from as phi i minus phi j or something like that. So, the evaluation of the gradient is readily is clear, when we talk about involving these two things. So, what about convective flux or convective or advective flux? So, this is where we have to be more systematic and more careful, because advective flux is something like rho u phi; and we know that this particular advective flux must be evaluated in the proper way that is not using central differencing, but using the fact that advective flux follows in a in the proper direction of velocity; therefore, we would like to use an upwind scheme for the evaluation of the of the flux here. So, and how can we make use of upwind flux, upwind thing here?

(Refer Slide Time: 47:00)



So now, we can say that this convective flux, which we can call as F C dotted with the area, is equal to the flux. So, essentially what we are looking at is the u dotted with the velocity vector dotted with the area vector. So, if the velocity vector dotted with the area vector is greater than 0; then that means, that flow is coming from this side, and the convective flux here is based on the phi i value, and the corresponding u value here; and if this is less than 0, so in which case the convective flux is based on phi i and u i. And if you dot is less than 0, then it should be the flow is coming from this side, and in which case, this should be based on (phi j, u j). So, in in that sense, we can make sure that the up-winding is is a honored here, and the sense of up-winding, where convective flux can go from left to right or right to left depending on the local velocity is is is evaluated in this way.

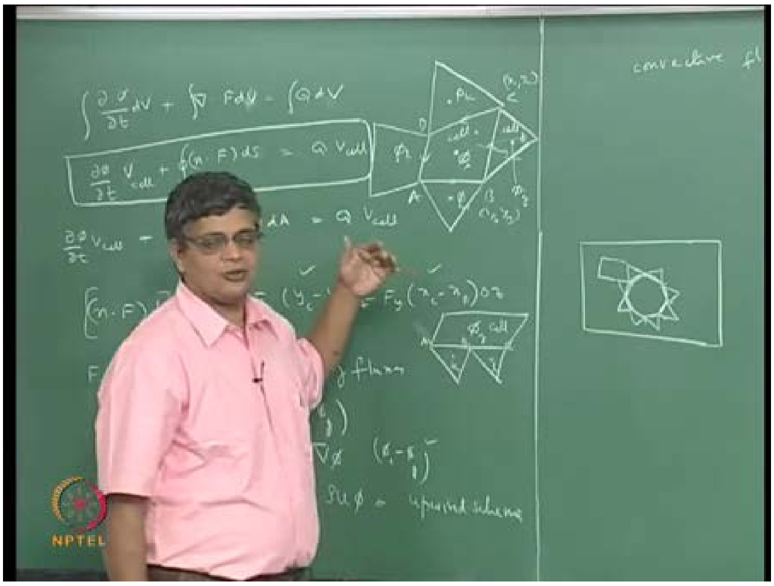
And so, this is this this is probably the simplest way of evaluating the convective flux in the sense of in in finite volume method in an unstructured method. And one can immediately see that if you wanted to evaluate the convective flux with second order or third order thing, where we would like to evaluate, we would like to introduce more number of points, then with this kind of spread it is difficult to it is difficult to make it more than first order. So, typically in finite volume method, we have first order evaluation of the of the advection, and second order evaluation of the diffusive flux. There are definitely methods, which have gone up to second order evaluation of the of the advection by reconstructing the direction from the existing values of the velocities and that involves much more work, but otherwise we can we can use this simplistic approach based on the local velocity information, we can either make use of… We we should always make use of the upwind up stream values of the variable in order to evaluate the convective flux.

So, that is with that thing we can have an evaluation of the flux as well as the areas, so that the total quantity of the of the phi, which is coming through the phases is evaluated based on the local parameter; and in in that way, we can make use of we can make use of we can ensure flux conservation, when evaluated for different different control volumes, which have that. And of course, it is one can also ensure the same thing by making sure that making some sort of account entry of the flux that is evaluated through this; and when we realize that, this cell has is connected to this cell, we do not make a separate evaluation the flux through this, we make this flux coming from this already has taken as evaluated, and then put it through this way.

And in cases like that here, one can evaluate the total flux leaving through this, and then based on some sort of area waiting, we can evaluate the… We can apportion the total flux leaving into parts, which are coming into this and this together. So, we have to be careful in the evaluation of the fluxes for different cells, so that whatever the flux that is leaving goes into, and whatever the flux that is going into the adjacent control volume are one and the same, and the evaluation of the areas in all those things are based on the vertex points; so that the areas are competed the same way.

So, if we do this, then the discretization of the governing equations, put in the conservation form using this approach becomes very flexible, and it can be used for any complicated n sided polygon, which would form the control volume; and we can take advantage of this to deal with any complicated geometry involving for example,

(Refer Slide Time: 53:08)



a circle inside rectangular block, which is preventing flow from going through. So, this cycle here can be made into number of sides like this; and now we can put control, we can put tiles here; and the interior can now we made into polygons and so on like this, so that using this triangular elements, we can make proper, we can take proper account of the shape of the of the domain of the flow domain; and it can be readily match with the anything else.

So, making uses of rectangular combination of rectangular and triangular elements in two dimensions, we can take account of fairly complicated flow geometry. And since the discretization of things does not depend on the coordinate directions and so on, it makes it easy for us to develop an equation for a particular value of the variable in a particular domain by consisting it by looking at all the fluxes. And this finite volume method also enables the boundary conditions to be incorporated in a natural way. If a flux value is specified through this, then that flux can be readily taking into account, when evaluating this. So, we do not make a separate statement, separate evaluation of the flux; the flux that is specified as part of the boundary conditions will be will be used, and evaluating the flux coming through that particular phase. So, the in that sense, that makes it possible.

And as a final note, if you apply finite volume method to a structured grid, we tend to do, we would get the same equations for the same degree of accuracy for as we would get using finite difference methods, except the only difference may be in the incorporation of the boundary conditions, there you might find some difference between a finite difference method in a finite volume method. Otherwise finite volume method apply to structured mesh is not going give us vastly different results. But when it is applied to this kind of combination of these tile elements, then the power of finite volume method comes through; and as mentioned earlier, if you want to do local refinement and all that, then doing this, using that in a finite volume method is is fairly straight forward. So, that is why the advantage of the finite volume method comes through.

And the overall puzzle of solving the flow for something like this is not yet over, we have looked at how to discretize the governing equation using finite volume method, and then convert it in to ah algebraic equation. But for this method to work, we should be able to cutout the overall flow domain into tiles, such that the complete available area is accounted for; and we make tiles which are not overlapping with each other or crossing with each other; so, that that sort of grid generation is very important, and that is what we need to understand before we can say that we are able to tackle the finite volume method or the solution method for a complicated geometry, which we will deal with that in the next lecture.