

CFD APPLICATIONS IN CHEMICAL PROCESSES

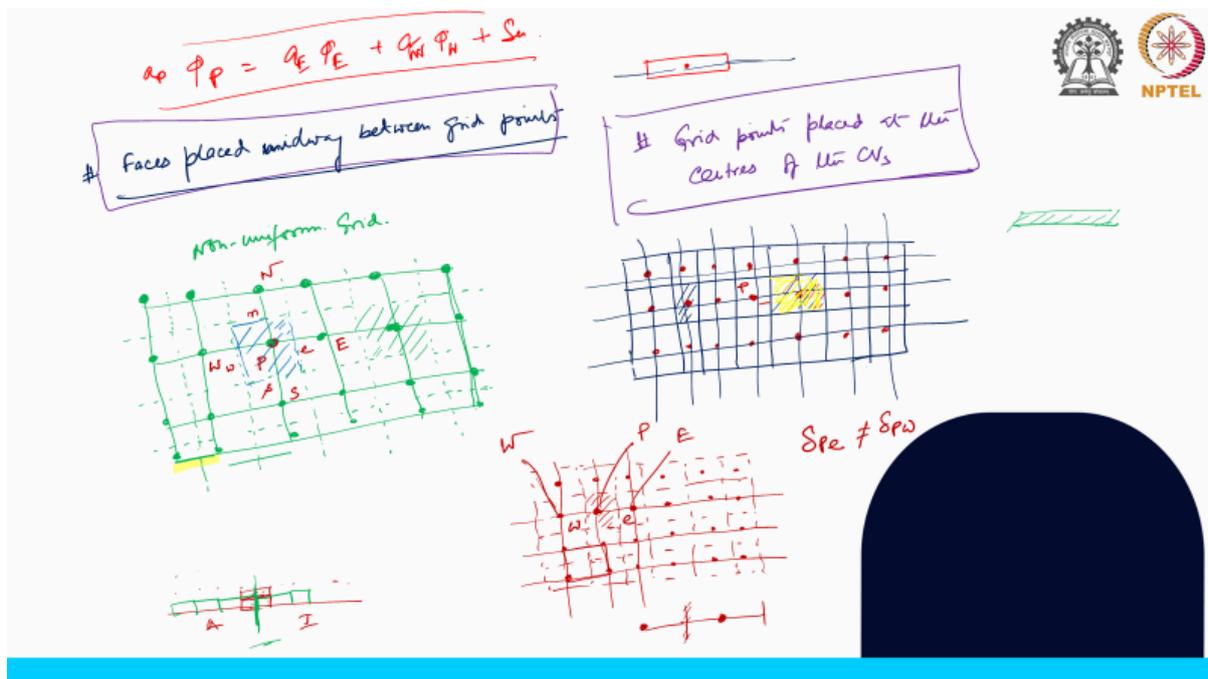
Prof. Arnab Atta

Department of Chemical Engineering
Indian Institute of Technology Kharagpur

Week-05

Lecture 21: Finite Volume Method

Hello everyone welcome back once again with another lecture on CFD applications in chemical processes we were discussing about finite volume method its intricacies its details in the last lecture in fact last couple of lecture we started looking into its implementation Taking couple of examples, we started with steady state conduction and then we added a source term on it with that and we have seen that how this finite volume method is implemented and how accurate that can be. when we compared the finite volume CFD prediction or the CFD method that is based on finite volume how that is comparable with the analytical solutions because all the examples that we have seen those are pretty trivial or simple and for those are the benchmark problem for those we had the analytical solution



Now today we continue discussing on that on the similar line because in the last lecture what we did we have seen the influence of diffusion and convection. So, steady state conduction which is essentially a diffusion equation and then on top of it we added a source term and in the last lecture we added some convective term with an example of the pin fin problem. Now, today we will work on it but before that since everything we are doing in one dimensional for the sake of simplicity and also to which is easy to understand. But what happens actually in the real world the most problems are 3D in nature that we have to implement this strategy. And in

those cases how the control volume looks like or what is the generic form of the expression that we have seen is $A_p \phi_p$ is equals to $A_p T_p$ in the previous cases.

that in most cases we tried to find out that instead of TP is very specific example for taking temperature. So, essentially $A_p \phi_p$ is equals to $A_E \phi_E$ plus $A_W \phi_W$ plus S_U . So, in this form we had the previous expression. or the previous examples for which we have seen that how the implementation happens and particularly for the one dimensional problem. But for one dimensional problem we have a simple control volume and we mentioned the strategy that either we can have

equidistant grid points or equal size control volumes or those can be of different sizes. Now, if that happens then the strategy I touched upon is that the first strategy of drawing the grids on a domain that can be is that face those are located or placed midway between the grid points. So, faces placed midway between grid points. So this is one strategy of drawing this control volume or to have in the case of domain say even in 2D cases the complexity appears that how do we place those grid points.

So or also even in case of the 1D cases but that is best illustrated when we have a 2D grid. Now the point is Faces are placed midway between grid points which means I draw the grid points randomly without considering that whether those are equispaced or not something like this. So, here what you see that I have deliberately place the grid points at non EQE distance or the distance between the grid points are not exactly same as that of the previous cases. So, for example, here we have a shorter length for this for this.

grid points or the mesh where we have a larger distance here between the two nodal points. So, this basically becomes a non-uniform grid. So in this case then the control volumes are drawn taking the strategy that faces are placed midway between grid points. So we take the faces with this dotted line considering that this is the midpoint between the two nodal points. So, considering these are the midpoints for each case.

So, in this case and then again in the x direction So, my control volume around say this point P if I consider that this is my point P the control volume looks like this. So, in this case this is my control volume, but if the point of attention shifts to say this point and in that case what you see that for that case my control volume is this. So, clearly the control volume sizes are not equal or the volumes would not be equal if it is in the 3 dimensions. Now, the point here to be noted is that this grid point actually is not at the centroid or at the center of this control volume.

But the assumption that we do during the previous calculation if you remember is that the P is the representation of that control volume. That the properties that are assigned at P is the representative value of this overall control volume. Now, consider the situation that where you

have a non-uniform temperature profile. So, in that case P truly does not represent this control volume because it is not at the center.

So, the averaging that we do or taking a value assigning P and considering that it represents that control volume that is not appropriate. So, that introduces some error in the numerical model if in this way the grid is constructed. So, P is basically not a true representation the values of P because all the scalar values that we assign in this grid or initialize are stored here and subsequently it is recalculated based on the boundary values. So, the point is that this is one way and in this case if I complete the nomenclature, if this is P , this point is E , this point is W . In 2D, now we say since this is also my nodal point or grid point,

we say that this is the south point and this is the north point. So this is the east face, this is the west face, this is the north face and this is the south face of this control volume. And similar to our previous strategy where we applied this discretization of the gradient across E and W is applied for north and south faces. So now the other strategy The other strategy can be that grid points placed at the center of the control volumes.

So, this is the second strategy. This is the one strategy and this is the other strategy by which you can draw grid. which means in this case when you saw while I have drawn this grid I started with the grid points and then I placed the faces at the midpoint between the two grid points. Now in this strategy what is done is that I draw my control volume irrespective of whether it is equal size or not. say I can draw the control volumes like this of non uniform size.

It is clearly that this volume and this volume are different, but here then what is done is that we place the grid points at the center of this control volumes. So, the grid points become this is at the center of each cell or the grid control volumes rather. So, these are the control volumes inside and in the center we place the grid points. Then basically this connection of these nodal points forms my actual grid.

So, which means here once I have. So, this extended lines is are basically the grid lines and in between what we have is the control volume. So, one control volume in this case is essentially The combination of this dotted that the shaded lines that you have seen I have drawn earlier here that this is my control volume essentially. So this is my say one control volume. This is one control volume. So in this case, what is the advantage?

The drawback that we had in this case that the cells are not at the midpoint, the grid points are not at the center of the cell or the meshes, here that drawback is overcome. That we work on the drawback by placing the grid points deliberately at the center of each control volume. but by doing so what happens is that the faces that you see here say here if this is my grid points. So, the distance between say if I consider my this is P point and this is so let me once again

redraw it clearly so that it becomes very apparent to you is that this is my control volume overall domain and I divide the control volume in non-uniform size

So, these are my control volume this dash lines and then I place the control wall this grid points at the center of these control volumes. Once we do that, we see that this is my actual grid. If I consider the connection between all these, so this is basically the grid and the control volume is essentially this all the dashed lines. the enclosure by 4 dash lines on the 4 sides. Now, say in this case, I have a point P here.

So, if this is my point P, this is my east point and this is my west point. One control volume is this around P. where you see that if this is my east face and this is the west face the distance between P and E is not equals to P and W small w . So, the central discretization scheme that we did while approximating the gradient. is not appropriate here because this grid point is not placed at the midpoint of two So, that approximation that led or the discretization scheme that we applied there considering this point are exactly at the faces are at the midpoint between the two grid points accordingly we estimated the flux at this faces are not appropriate.

So, both has their drawbacks both strategy. In this case the center is the centroid is not the grid point in the case of non-uniform grid when we start with the grid point first. when we start with the cells or the control volumes first and then place the nodal points at the center of it. We overcome this drawback of the point nodal point truly representing one control volume, but then the estimation of flux at the faces are not appropriate by the central differencing scheme that we used earlier. that introduces error there the numerical error in such strategy.

So, which one we should choose the question comes or what should be the strategy then. So, then that is why there is no such uniform or one size fits all kind of a scenario. So, what happens here based on the problem description, based on the boundary condition specially this boundary condition sometimes dictate that what kind of meshing strategy or the grid strategy you should adopt. In the case of uniform grid or the uniform size meshes both the strategy results in same

number of grids and same I mean all the drawbacks are there is eliminated in the case of uniform grid. But several times that uniform grid is difficult because if you have irregular shaped body this uniform grids are difficult to adopt or to implement. So, that is why this boundary condition dictates sometime that which kind of grid we should Now consider that your face your one boundary is having half of the boundary is adiabatic and a half of it is isothermal. Now in that case if you draw the grid randomly

draw the grid points randomly and then connect it and then you draw the control volume. The chances are that one control volume may share at the boundary the nearest to this that overlaps with the boundary face. The chances are that you do not have control on this boundary on this particular mesh or the cell where it would share both the boundary conditions. And that creates

discontinuity in the problem. Because you have a clear jump, it may not be exactly midpoint or somewhere, one cell at the boundary or multiple cells on the boundary can have this overlap.

For 1D, it is 1. For 2D, it will have many cells like that and 3D cases. So, in such case, if you draw the cells at first carefully understanding that, okay, I have my boundary condition, one boundary condition till this point. So, I have my cells here like this and the other cells. So, the clear demarcation of the cell phase that also coincides with the boundary condition, the different boundary condition, then you eliminate that chance of discontinuity.

Which means in this case the strategy should be that you draw the control volume first coinciding with the boundary condition discontinuity point and then place the nodal points at the center of it. The point is this commercial CFD solvers give you this option. and again without knowing you may be using but the point is there are options user feed options that whether you have to do the edge meshing first and you develop the grids or there is an option that how many grids you want in that case in the in the computational domain so those two actually incorporates these two strategies If you do the face meshing or the edge meshing at first, edge means a single line, face means in the case of 2D, this complete face. So, if you do that, this face or edge meshing first, you have a clear control on such kind of a problem where there is a discontinuous boundary condition or a different boundary condition on a same boundary.

So, this is the intricacy or these are the points that are there in the commercial solver like ANSYS Fluent and all where you have to be careful while choosing it. But the point is all these drawbacks can be eliminated or say I would say not completely eliminated but can be reduced the error that it introduces from the both strategies if you have very fine grids. if you do finer meshing then these problems are possibly eliminated or you reduce it to an extent which is say within acceptable accuracy limit. So, I hope you understand this point considering this the strategy you have to adopt that what kind of problem is given to you. and for that what should be your better strategy if you would like to have the more accurate solution, if you want to achieve more accurate solution even with the coarser grid.

Because as I you now understand even if I just say that the number of how number of control volumes increases your computational time because for each control volume you have to solve those discretized equation and number of increase in the control volume or the grid points increases those number of equations and accordingly that increases the computational demand. So sometimes we start the problem solution with a coarser grid to see whether the idea that we are having the proof of concept idea is working or not. If it is having the similar physics or the trend of the result is same that we are expecting or that is the actual case then we go for the finer meshing.

And then we achieve more accurate solution. And once the model is validated, we go for prediction for a wider operating range that has not been covered experimentally or the solution is not available for such complex problems. So with this I will stop here today that the meshing strategy we discussed in this brief class and then in the next class we will move with this finite volume further where we will take up the diffusion and advection problem because we have seen only diffusion we have seen diffusion plus convection and then now we will see diffusion plus advection. So thank you. Thank you for your patience.