Open FOAM Lecture 2:

Hello everyone. Welcome to the lecture series on the course “CFD using openfoam”. This is the second lecture which is based on overview of CFD methodology. The course is brought to you through FOSSEE IIT Bombay. I am Sumant Morab, the instructor for this course. Along with me, professor Janani is the coordinator of this course.

In the first video lecture, we saw what is CFD and why it is necessary to study CFD. We tried to compare CFD with a video camera like tool. Later we saw how open foam acts as a tool to study CFD and the flexibility which open foam provides for CFD investigation. In this video lecture, we will briefly look at various steps involved in the CFD study

The outline of today's presentation is as follows. First, we will look at the various steps which are involved behind each CFD study. we will study Pre-processing Which involves geometry creation as a first step. Then we will look at how we can convert the mathematical equations in a format which can be Solved by a computer. After getting the equations, we will look at how We can make the Computer To solve those equations using a solution scheme. Finally, we will look at how we can use the Results obtained to convert them into a format which is more understandable in real world and how we can test whether the solutions obtained are correct or not.

The following flow chart summarizes the steps involved in CFD study. First, we create the domain and generate the mesh in the Corresponding domain. In the second step, we try to convert the equations which are usually partial differential equations into form which is known as linear algebraic equations. Basically, which can be solved in a computer program. This procedure is mainly called as discretization. In the third step we select the solution methodology. After finding the results, we try to get the necessary parameters out of the results. In the final step, we test the obtained solutions by using either experimental data or by using the data which is available in the literature.

Coming to the first step, which is domain creation. So, whenever we want to study any problem, we must understand what is the area in which we want to Calculate the variables. For example, if we want to study flow over a car which is moving on a road as shown in this figure. Now imagine that there are some birds which are flying at a very large distance from the car. Now whenever you want to Perform CFD investigation, you have to select an area around the car such that outside this area, the flow features should be normal and must not change in a larger quantity. So, if you see this particular example, we can say that below the road there are not much flow features which can be observed. Even the bird which is flying, doesn't create much disturbances on the car surface. So, we can safely ignore that and the domain of interest can be considered to be the area Behind and in front of the Car As shown in the second figure. In the second example, we see flow inside a channel Which is bound by top wall and bottom wall as shown in this figure. Now the walls are assumed to be rigid in nature I.e, they do not move from their position. In such cases, we can safely Remove the wall and replace it with a Zero velocity boundary. Also, we notice that if we divide the channel Vertically into two parts. The solution remains symmetric about the center line. That is, the results which are obtained on top half part Become the same in the bottom half part also. So, we can consider only top half part for calculation. In this way the final domain becomes reduced and is shown in the figure given in the right side.

Now we looked at how we can select domain for our study. If we see the open form implementation. We can Find the Block mesh dict file In the System folder In the general Structure of the open form problems. For example, you can go through this directory given here which is tutorials/ incompressible /ico-foam and then cavity. Once you get into this cavity folder, you will find three different folders. The zero constant and system. In the system folder, you find a file known as block mesh dictate. The geometry can be modified through this file. So here you get a set of vertices where in the Coordinates of the Domain needs to be given. For example, if you are creating a Cubical box as shown in this figure, you need to define the eight vertices. If. The vertices are started from the origin, which is denoted by zero. Usually, an anticlockwise orientation is considered as shown in the figure. Now once we define The vertices From 0 to 7, We must define the corresponding faces of this cube. For this, we can use the right hand thumb rule wherein The thumb has to be pointed towards the outward normal direction as shown in the figure. The direction in which the fingers curve gives us the direction in which the vertices should be considered for naming the surface. So for example, in this particular case, If we start from the vertex 5 for the bottom face, then we must go on with 5401 And not 5104. Similar Thing can be done in all the other five faces.

The second step involved in the CFD study is the mesh generation. We saw in the last week that in a video camera, the color was obtained due to the intensity and the wavelength of the light which is falling on each pixel of camera. In CFD, the color graphics are mainly obtained by solving equations on each pixel, which is known as control volume. Now the process of generating these pixels is known as meshing. In this figure you can see the problem which needs to be investigated. It is a square cavity Inside which there is a flow generated due motion of top lid. This domain can be divided into small number of parts as shown in the figure given on the right side. Now each control volume, which is indicated by the blue color, has a central point which is called as the centroid or the internal gridpoint. Similarly, the blue dots indicate the boundary Gridpoint. Now we have to solve the mathematical equations on these Grid points to obtain the final picture. There are two ways in which the mesh can be generated as can be seen in this left figure. If we consider unstructured mesh, then the control volumes are arranged in an arbitrary manner and a connectivity Is not fixed. These kinds of unstructured Mesh are mainly helpful when the geometry of the problem becomes very complex. In open foam two different Options for mesh generations are present. One is the block mesh & the second is the snappy hex mesh. So basically, in hex mesh here Creates hexagonal control volumes. And to fit into the complex geometry, usually the hexagonal domains are subdivided into many other meshes. In case of Simple hexagonal block mesh Which can be Generated using the simple hex command as shown in this figure. So, for example, if We need 10 points along X direction and Y direction, we can simply write the vertices of the Block where in the mesh needs to be generated and then indicate the number of grid points using the simple grading command.

Now the important step in any CFD investigation Is the algebraic formulation of the partial differential equations. After creating control volumes through meshing procedure, we have to solve equations on each and every control volume so as to obtain the values of variables. That is the velocity and the pressure of the fluid. Now if we see the equation, They are usually partial differential equations as shown in this equation #1. So, can we solve the following equations directly on a computer? Definitely these equations cannot be solved directly. But if we see some other set of equations For example, the second set of equations given here. These equations Can be solved through a computer using direct inverse or through some other indirect methods. So, the objective here is to convert That set of equations given in one into a form given in two. So basically, we have to convert partial differential equations into linear algebraic equations.

For this conversion of partial differential equation into linear algebraic equations, we have mainly 3 methods. So, the first method is the finite difference method. Let us consider a simple example Du/ DX is equal to three which needs to be converted. Now we have the Grid points denoted by W, P, E. If we want to obtain the linear algebraic equation on the grid point P. We used the Second order central difference scheme wherein we write the differential du by DX as the Difference between E,U at the earth grid point and W TH grid point and the distance between East and West Grid Point which is 2 times delta X as denoted in the figure. So directly we kind of get an equation. The similar exercise can be performed on all the other grid points to obtain mainly 4 set of equations for solving 4 Values of variables in the grid points. Now this can be solved through different approaches which are which will be discussed in the upcoming slides. The second method involves finite volume discretization wherein we Kind of integrate the differential equation over the total volume of the control volume. And then we divide the whole integration by the volume of CV As shown in equation 4. Applying Gauss divergent theorem, we'll end up with surface integral of U Which can further Be Calculated as the difference in velocities. Thus, we'll end up with the same set of equations which we got from finite difference method. The third method is the finite element method, wherein similar to finite volume method, we divide the Domain into small number of elements which are called as finite elements. And the main objective in this particular scheme is to Obtain an approximate solution for U such that the residual R, which is du/ DX - 3 Becomes Minimal in nature. With this objective, if the algebraic formulation is found out. So, these are the three main methods and the differences which exist among the methods.

Now the 4th step is the solution methodology. Once we get the linear algebraic equations through any of these discretization schemes, The main Task is to make the computer to solve these equations, as I said earlier, one approach is the direct approach wherein we kind of calculate the inverse. To get the values of variables as given in this equation 11. So, if we have a Matrix equation in the form of Ax is equal to B. We can write directly X is equal to A inverse B. this A inverse B can be calculated by the computer very easily. Now, this is very useful for matrices Which have very small dimensions. But if the dimension becomes very large, that is when we have large number of Grid points then this becomes a very expensive method. In in those cases, second approach which is called as iterative or indirect approach is used. Here we guess an initial value. Update it iteratively till the difference between iteration becomes close to 0. In such a way we try to solve these linear algebraic equations. You can refer to mainly 'Gauss-sidle' method, Gauss Jacobi method online to know more about the indirect approach. But these will be discussed in the later lectures. Now if we compare the direct and indirect method, the direct method Basically gives the exact results of linear algebraic equation, whereas the approximate solution of the linear algebraic equation is obtained with indirect method. Also, direct method becomes computationally expensive for large number of variables, whereas an approximate solution can be obtained with less time in indirect method. It is also observed that the direct method becomes unstable in some of the cases where Nonphysical results are obtained, whereas feasible solutions are obtained in the indirect method. Example for direct method involved direct inverse calculation or Lu decomposition of a matrix. In indirect method we have Gauss Seidel, cost Jacobi etc. In the open foam, indirect method using smooth gauss sidle or the conjugate gradient is implemented usually.

Now the last step involved in a CFD investigation is the post processing. Till now we saw how to calculate the flow variables starting from Domain creation, mesh generation, Converting the equations and then solving them through different solution schemes. Once we are able to calculate the flow variables like velocity, pressure, temperature. What is the next procedure? To understand the effects, usually we have to see some scientific or engineering relevant parameters. Because the Fluid flow parameters directly do not give a good meaning to the problem. For example, if we consider flow over an aircraft, it becomes very important to know the total lift force on the aircraft body so as to decide the load that can be carried on aircraft. Now this lift force is calculated using the total integral of the pressure over the surface of the aircraft body. So integral operation needs to be performed here. And the lift force can thus be calculated. This is one typical post processing parameter. Now, in the second case, if we consider flow inside a tube wall, shear stress needs to be calculated to check the probability of wear and tear of the tube walls. Now this wall shear stress is obtained through the tangential velocity gradient along the perpendicular direction of the tube. So, in this sense we are using the flow variables to calculate some external parameters which convey a very broader meaning. In open form. We can. I rectally calculate these post processing parameters by modifying the controlled file which is present in the system folder. So, as I said earlier, if you go into the any test case folder you will find Zero constant and system folder. Now in the system folder we already discussed the relevance of block mesh DICT file. If you go to the control dict file you can add some functions as shown in this figure to calculate the post processing parameters. Also, you can also calculate some parameters on the fly, that is, after the calculation of the solution. So, after the calculation gets over, we can enter the following command which is given here. That is the solver name which we are using post process function and the post processing parameters. So far here it is Wall shear stress. You can modify it so as to directly get the necessary postprocessing parameter. Now the most important aspect in a CFD investigation is to test the solutions. When a new problem is attempted, it is important to check the results. For simplified geometry first with those available in the literature. So, there are two approaches through which you can test. The solution which you have got through open for the first approach is the experimental comparison. If the previous literature contains experimental results, direct comparison can be performed or the experiments can be performed on our own If it is really necessary. The second approach involves comparison Of the current results with the previously published numerical literature. Especially it is A good practice to compare with the three-dimensional numerical simulations. So, to summarize, we saw the following steps which needs to be carried out in a CFD study. First is to create the geometry, then took geometry to Create the mesh that is the small control volumes which acts as a pixel. Then we go on to discretize the mathematical equations Which are usually in partial differential form in two linear algebraic equations which can be solved through a computer through some solution scheme. After getting the results, we saw how we can extract some parameters from the flow variables to get a better understanding of the changes occurring in the flow. Finally, we saw how we can test the solution which we have obtained through Either previous numerical literature or experimental work. These are the references used in the current Presentation.

Thanks for listening.