CFD using Open FOAM : Lecture 1: Introduction

Hello everyone. Welcome to the first video lecture on the course of CFD using openfoam. This lecture is dedicated to the introduction and a brief overview of the subject. The course is brought to you through FOSSEE IIT Bombay. I am Sumant Morab & I will be the instructor for this course. Along with me, professor Janani S M is a coordinator for this course. The outline for today's lecture is as follows; first we'll briefly understand what is CFD, we will try to get a better appreciation of CFD through an analogy with a video camera. After understanding the subject, we will try to see what is the importance of studying this subject. We will consider an example of a biomedical case to understand its importance., finally we will see how open foam acts as a tool for performing CFD investigation and why it is better than other tools available. Before starting any subject, we have to first understand what the subject is about and how it works. CFD stands for computational fluid dynamics; basically, we take the help of computers to study the fluid dynamic phenomena. By fluid dynamic phenomena what is being meant is you want to study the force which the fluid impacts on surrounding structure or how it moves in a given area. understanding this will basically help to design better systems. So, we are using computers to understand this.

To sum up, whatever experiments you performed in a laboratory setup the same fluid dynamic experiments are being performed on a computer in CFD subject. Now let us try to get an answer for the second question which is how CFD works. does CFD mean only applying computer programs for performing experiments. No. that is just one part of CFD. Like experimental setup, we have to write the programs define the geometry of the area which we want to study. Once we do the experimental setup and see the fluid motion using the CFD theory, we have to also calculate some important parameters which can give us an indication of the motion in flows which is a very important step. now let us try to understand a CFD using a video camera analogy. you all know that video camera takes pictures at different instants of time in a particular region and basically clubs them together. OK so for an example you can see in this picture that the camera is taking the pictures of water flowing in a river bed. The intensity of the light falling on the lens of the camera helps to get the picture of this water flowing so this is in particular for video camera. now what happens in CFD is that we again calculate the changes in fluid velocity, the amount of energy which it carries at different instant of time in a particular region of interest and we basically club all these images and play them as an animation. one thing which you have to notice here is that these images are actually got by solving the equations in separate pixels. Like a camera, instead of the light which gives the picture here the mathematical equations give us the picture of fluid flow. so that is the difference. also, CFD is a silent video camera we cannot extract sound. now as we have got a brief understanding of CFD, let us see what are the steps involved in any CFD study. first step is the development of the computer program using any language of your interest so this process is similar to script writing in a movie production. so, before the shooting begins the script is written and finalized first. so, in CFD also we take the equations and write the computer programs to solve those equations in the second step we use the written scripts for a particular problem. now for example if we want to shoot a movie of air flow over a cylinder basically take the script run it for this particular case OK. so, this is somewhat similar to shooting in a movie shooting process which happens in a movie. the third step involves analysis of results wherein once we get the fluid variables like velocity the pressure through the equations which we solve & then try to make them in a more understandable way through some parameters. also, we make scientifically or engineering exciting movies so that the reader can understand it in a better fashion. this step can be compared to the editing face of a movie where in some extra graphic affects like VFX are added so this particular process is somewhat similar to the movie editing. so as far as now we learned what is CFD and basically the steps which are involved in our study .now an important question is why should I study CFD & what is the practical relevance of this subject. we will try to understand it using two situations. in the first situation consider that you have three designs of Formula One cars and you want to select one out of those three for a final racing competition. now you have to make a decision based on performance of the three designs which you have no the first option is that you actually perform experiment on the prototypes of the Formula One car. you actually get a small prototype manufactured and you conduct flow experiments in a wind tunnel and then calculate the total force which acts on the car and try to say that one particular design performs better than other. this process as you can clearly see is costly and also time consuming. so instead of this you can perform a study by just sitting in front of a computer you generate the cad geometry of the three designs of cars which are available and basically take a particular region of interest where you want to study the air flow and then solve the equations in that domain. you directly can understand which design is better within a very less time. the second situation involves cases where you just cannot perform experiments. in those situations, CFD is the only way out. nowadays industries ranging from food processing to aerospace and also in biomedical field are using CFD to obtain higher profits and they really need food CFD engineers. so, in this sense we can understand that CFD is really a very important subject to learn especially for four industries. now let us try to get a better appreciation using biomedical case study this particular case study corresponds to the second situation which we saw earlier that you cannot just perform experiments in some cases. so, this is a such a case. a person has a blockage in the blood carrying artery and basically a stent needs to be put in the artery to normalize the blocked region. so, two designs are currently available and our objective is to avoid re-stenosis; that is blockage after sometime. so, we have to select stent in such a way that it gives a correct treatment for the blockage. OK so these are the two designs which are available. now since this involves artery in the human body you cannot perform experiments. you cannot use in such a material and develop such a prototype because the variation in the design is very high. so never you can approach that particular design of the artery in reality by manufacturing. we can extract the geometry from the MRI scan and perform a CFD analysis on the geometry extracted and study vital parameters and see which design performs better I.e, in which design of this stunt performs better. so, as you can see in the figure both stents are inserted one after the other and then the CFD investigation is performed .so this kind of study can be really helpful in saving patient lives. now whatever we had discussed we saw that we were studying fluid flow which involved only one particular phase but CFD is not restricted to only one particular phase of fluid flow process. you can also study multiphase flows where there are two different phases of material basically gaseous and liquid state. These kinds of problems find applications in food making to even Bio medical field. the second aspect involves with heat transfer study so apart from fluid flow variables we want to calculate the temperature in the area of interest. through this we can actually predict the changes in the weather patterns very easily. because the airflow around the ocean bodies along the land surface greatly impacts the variation in the temperature.

now we saw what is CFD and why we have to really learn. open foam is a tool for CFD investigation. now whenever you want to perform a CFD study you have two options; the first option involves developing your own software by using the governing laws and then basically converting them into it a form which can be easily solved by a computer. the second option is to use the solvers actually developed by others. you need to only set up the geometry of the program, give proper boundary condition and the program parameters like the viscosity of the fluid the density of the fluid, domain lengths and then just run the case. so, for the second option there are many softwares which are available like ANSYS, COMSOL, ADINA etc but all these softwares are not open source in a way that they need to be purchased. open-source software is meant for this purpose. open FOAM stands for source field operation and Manipulation is basically a C++ tool box for mainly development of customized numerical solvers. you basically can also develop your own solver in this tool box OK. and then also you can do the pre and post processing. you can create the geometry and that is preprocessing and also you can extract engineering or scientifically relevant parameters like the force which is acting the stress which is acting on the solid body which is in contact with the fluid. if we put it in simple words open foam is a book which contains several programs which we can directly copy and use or add it to suit our problem. it basically contains several numerical programs you can directly use that program or else there are some blank pages. also, there are unlimited blank pages where you can create your own solvers in those blank pages .so if a particular study is not already available in open form for example acoustic module wherein, we solve for sound generation sound propagation you can actually contribute towards the development. we saw what this open foam is. let us try to understand why we have to consider open foam for CFD investigation. now as I told you there are many commercial softwares with GUI and also you can write your own code only. but these both the capabilities are provided at a one particular location it is called a Open Foam. so that is why open foam is the best option you kind of get a software like feel; very directly run the code also and if you want to make modifications the code is already clearly available to you; can edit it and modify it you can add some files very well making it something new. so, these options are not available with other commercial softwares so that is why open home is the best option. also, apart from this case close you can also score the metric caseload in transfer as I discussed earlier. also, there is a module for fluid structure interaction. so, these aspects are covered in openfoam .

now let us summarize the contents of today's lecture; first we saw what is CFD subject and then we tried to get a better appreciation this subject by considering analogy with video camera. after understanding CFD, we saw what stages are involved in the CFD study. after getting an overview of subject, saw applications of CFD very important industries and in which kind of situations. finally, we saw open form as a CFD tool and water is capabilities and why we have to mainly focus on openfoam for CFD investigation I.e., advantages which the other softwares or the solvers do not give us. In the next class we shall see a brief overview of three different steps already discussed. we will get a more detailed description in the next class. these are the references which are used for today's lecture series. First mainly the contents are obtained from the introduction to development application analysis book. thank you for listening.