Introduction to Finite Volume Methods-I

Prof. Ashoke De
Department of Aerospace Engineering
Indian Institute of Technology, Kanpur

Lecture – 01

Welcome, myself is Dr. Ashoke De. I am currently working in the Department of Aerospace Engineering as associate professor at IIT Kanpur. Now, what I am going to teach about the finite volume methods? And the final volume methods just to give you an brief introduction, the final volume method is commonly known as one of the widely used CFD tool. CFD means the computational fluid dynamics tool. And this is widely accepted not only in the academic community, it is also accepted in the industry; The reason being that CFD now with the advancement of the computer, CFD has emerged as an alternative tool to the experiments.

It is cost effective also you can get the solution in very quick turnaround time. Because sometimes also the experiments cannot provide you the information (Refer Time: 01:19) locations or inside the flow domain. So, that makes it very difficult to analyze a complicated system or rather realistic problem. In that sense, CFD has become the very, very effective tool which can get you the solution in quick fashion. Now, there are different kind of CFD methods, one is very commonly known which is finite difference method, then finite volume method and the finite element method.

But in this particular course we will be talking about the final volume method, for each of these different techniques they have their different way of doing the things. So, essentially we only talking about we will be talking about the finite volume method. Now, in this particular course so, as you see what CFD does? You have a practical problem of interest, which you are interested in that could be a realistic problem like flow over your automobile vehicle for drag reduction, flow around your aircraft vehicle. So, it could be multiple things or flow through your air ventilations or the AC duct.

So, there are plenty of problems which are practical interest, and to look at the problems in details the measurements are not possible. So, in that sense CFD is applied. And what it does when you have a practical problem of interest. You first try to identify what are the constitutive relations or the governing equations that govern that particular problem.

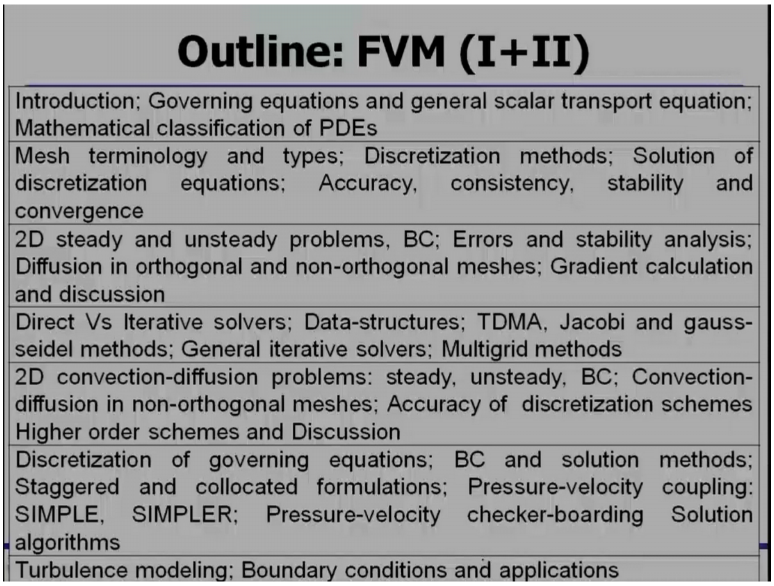
So, the physical problem either fluid flow problem or heat transfer problem then you figure out the set of governing equations.

Now, once you get the set of governing equation, these are essentially the partial differential equation. So, these partial differential equations actually provide the solution for that practical problem. And how, now this representative partial differential equations now needs to be converted to the linear system. And this process when the PDE's or the governing equations they are transformed to the linear system in between you apply your numerical technique; that is, for this particular context of this I mean lecture is the finite volume method. So, you apply your finite volume method and that set of governing equations it transformed to the linear system.

Now, you apply your linear solver. Essentially the linear system, one has to know the properties and find out the solution to the linear system. Once you find out the linear system that becomes the representative of the solution to the physical problem. So, that is what essentially CFD does. And as you can see that when you are converting your governing equation to the linear system your numerical technique comes into the picture and that is very much particular technique that is finite volume method. And this method we are going to talk about in this lecture series. And finite volume method just to note you this is again as I said it is widely accepted not only in academic community it is in the industry because, this has become a sort of a design tool for most of the industry.

Now, finite volume method essentially covers lot of topic. And this will actually in this particular class or the course it will be divided into 2 parts. One is the part 1, which is introduction to finite volume methods 1, and then introduction to finite volume method 2. But, if you look at the complete syllabus then you see this is the complete syllabus that you have for the finite volume method which includes your part 1 and part 2.

(Refer Slide Time: 05:07)



What we start with we start with all the governing equations, that is the transport equation or general scalar transport equation. And the classification of PDE's; that means, you talk about all this governing equations which will represent the particular physical problem, and you then understand the general transport equations and also the classification.

Then when you move along the process so, that is where it becomes discretization processes. Discretization process means your set of governing equations is now getting transferred through this discretization process to the linear system. And there you come across different things, like mesh terminology types solution discretization equations, now once you discretize the system; you have accuracy consistency stability convergence. So, these are the things we will talk about. Once that is done; that means, you have a set of technique which actually kind of confirm this discretization process then you move to the simple system of 2D steady and unsteady diffusion problem.

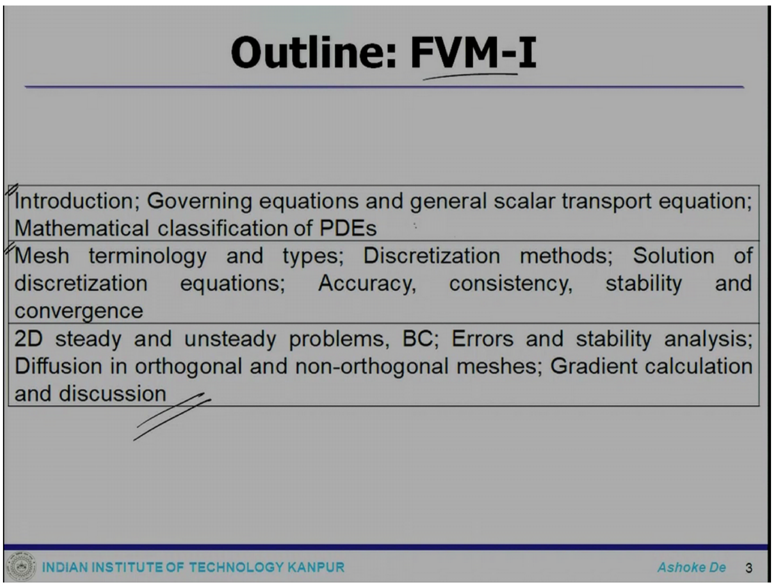
So, first we will look at the diffusion problem, and we will talk about all other associated thing which will be kind of associated with the diffusion problems like errors stability orthogonal non orthogonal system gradient calculations. So, once one understand the diffusion system, actually after discretization of the diffusion system it will lead to a linear system. So, one has to understand the linear solver. So, in this portion of the topic, we will talk about all the linear solver; that means, either direct solver or iterative solver, then we talk about different data structure tridiagonal matrix solver all other associated different solvers including multigrid method.

So, that actually sets you for solution of the linear system. Once that is done, then we will complicate the system and move one step further, talk about the 2D convection diffusion problem. In that case also, we talk about steady and unsteady we talk about different boundary conditions, we will talked about orthogonality non orthogonality, then accuracy and higher order scheme. Now, this becomes very important when you talk about the convection diffusion problem.

Having said that, there one is ready for moving ahead and look at the fluid flow problem. Now, fluid flow problem will be governed by pretty much Navier-Stokes system; which is a mixed partial differential equation. And it is complicated because now you have pressure gradient term sitting. There you talk about boundary condition, different solution method, then different pressure velocity coupling.

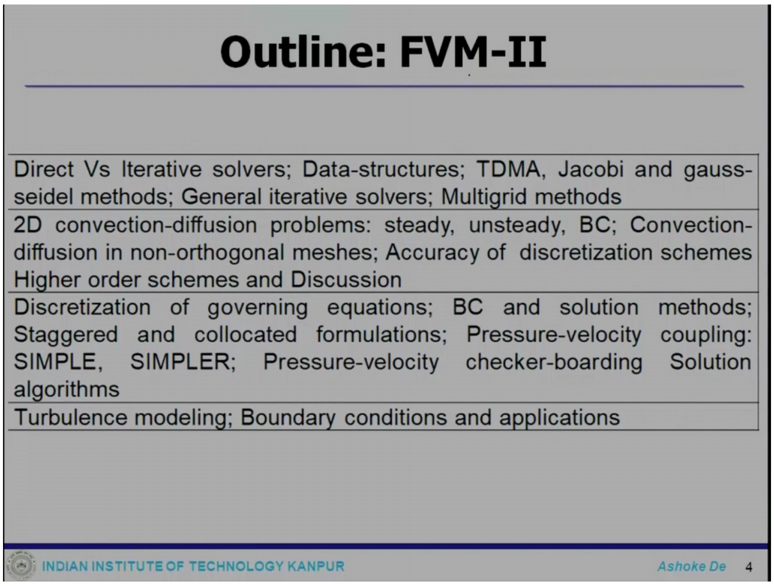
And this is a particular method where we will be talking in primitive variables that mean, pressure velocity density like that. And finally, towards the end we will talk about some advanced topics like turbulence model in boundary conditions and certain things. So, that is essentially the total lecture content of finite volume method. Now, for this particular part of this things this is divided into 2 part, one is the finite volume one which will be the content of this first part and there we will talk about introduction.

(Refer Slide Time: 08:29)



We talk about the discretization technique mesh topology and accuracy, then we end at the 2D diffusion system. So, the first part we will tell you about all your initial governing equation discretizations and the 2D diffusion.

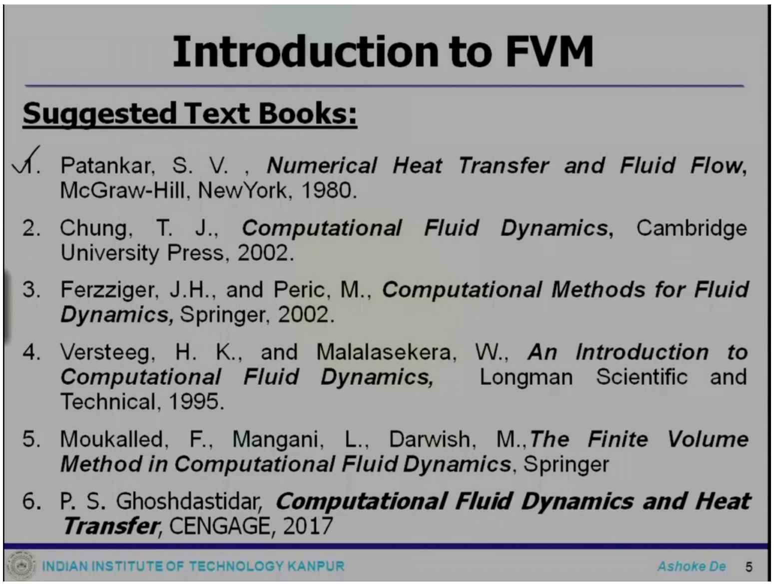
(Refer Slide Time: 08:53)



Now, in the follow up semester the second part would be thought which will talk about or start with the linear solver the direct and iterative solver, convection diffusion problem and discretization of the governing equation for the fluid flow problem, and finally, these advanced topic. So, at the end of these 2 lecture series that finite volume one finite volume 2 one should be able to understand clearly what finite volume method is, and he should be able to program on his own to solve simple problems. And then further their knowledge can be extended for the practical complicated problem.

Now, for getting into finite volume 2 one of the primary prerequisite would be the finite volume 1. As you can see this whole content has been divided into 2 equal parts one cannot do the course finite volume 2 without having a knowledge of finite volume 1. Because when you talk about lot of things in the diffusion system you may skip those things in the finite volume 2 part.

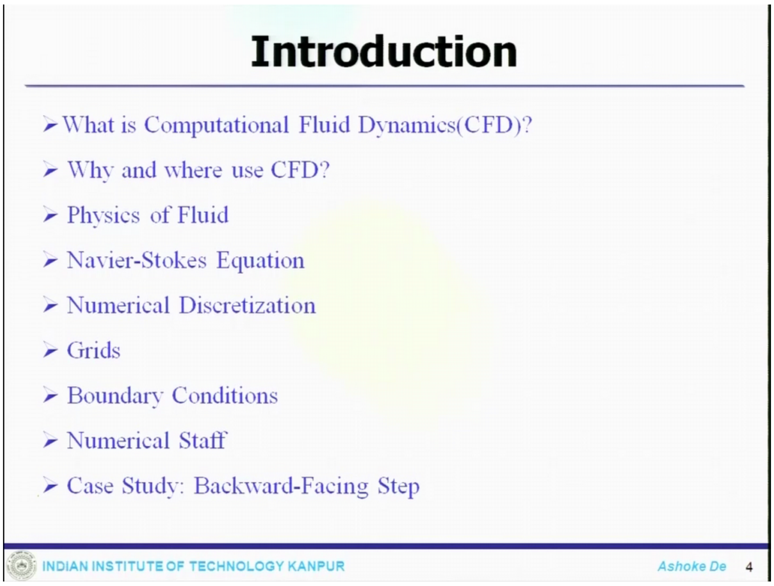
(Refer Slide Time: 09:56)



Now, the suggested text book the one of the very classical, and phenomenogical book is the Patankar, which talk about the finite volume method and one can cause I mean look at this book as a reference book. Another book which would be very good is the ferzziger book, this could be also reference book, but these book fourth and fifth these are could be the. So, 1 4 and 5 this could be sort of text book and one can see the because the material which will be covered this would be from all these 3 books.

And other books are for your reference. So, one can look at these books and get the complete course material. So, as you see that the total course has been divided into 2 equal parts one is part 1 and is part 2. And the first part will be talking about the first portion starting from the governing equation to the diffusion problem, and the follow up lectures in the second part or the follow up semester this would be part 2 where you start from the linear solver to the fluid flow problem. And this is what we will be doing in the complete finite volume series.

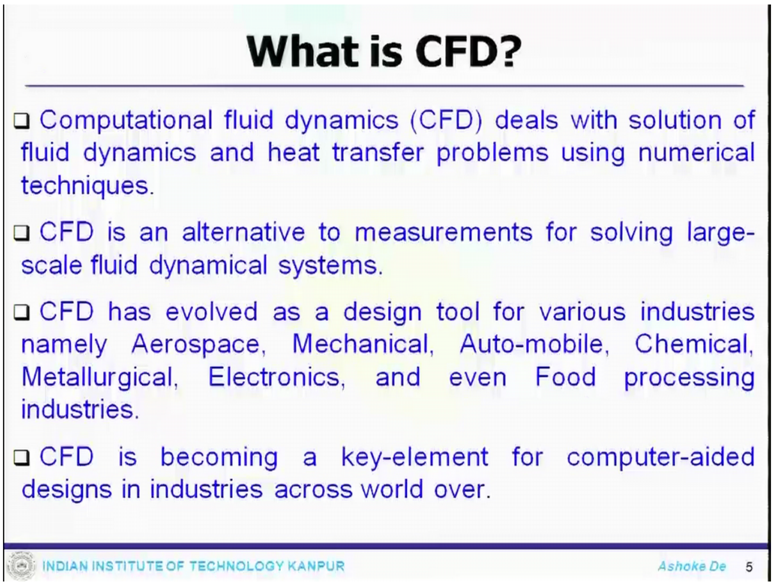
(Refer Slide Time: 11:00)



So, just to get you start at what I said that we should first know what is CFD all about and why do we need to use CFD. So, as I was saying so, first thing that we have to talk about what is CFD. Then, obviously, immediate questions come where do we use CFD and why do we use CFD, what is the advantage. Then how the CFD actually replicate the physical process of the fluid flow problem? And that leads to some sort of a equations like Navier-Stokes equations.

And once you talk about that that will lead to the discretizations of this system, and once you discretize the system, you get a solution of this system through the grids. And once you have these then boundary condition and other numerical issues that we will talk about. So, introduction lecture would take you completely to the background of CFD and this. So, first let us start with what is CFD.

(Refer Slide Time: 11:59)

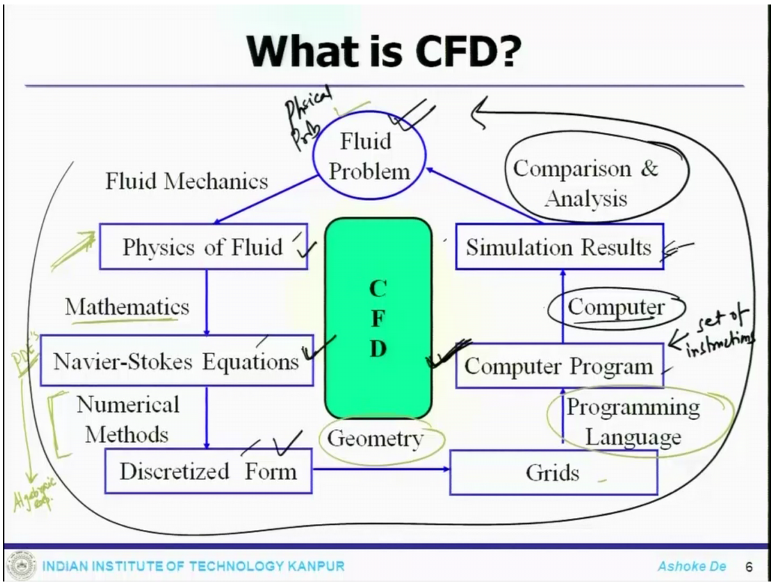


Now, if you look at it what is CFD. CFD is essentially deal with the fluid flow problems, fluid flow and heat transfer problem through some numerical techniques. So, that is what we call it a computational fluid dynamics; that means, the fluid dynamical systems which are exposed to the physical processes they can be represented through certain numerical techniques and that is all about CFD.

Now, CFD has been emerged as an alternative tool for large scale problem. And also the utility of CFD has been spanned over starting from aerospace system to the mechanical system, automobile system, chemical, metallurgical so, all these different areas whatever we are talking about here, here you can see the applications of the CFD. And we will get you some example of this kind of systems where people can use CFD, and how that is powerful compare to the experimental techniques, and how efficiently we can use CFD to come out with a solution to the fluid flow problem.

Now, essentially at as of today, CFD has become a key component or key element across the industries for design purposes. What I am saying by that is that now any design see 2 fabricated system and get a experimental done is going to be not cost effective; to reduce the cost so, people prefer the CFD or this numerical kind of technique. Once you use the numerical kind of technique, that get you the design data very quickly and using the design data you can fabricate your system and then get your experiments done and finally, go for the final production which will not only be cost effective also less time consuming. So, the turnaround time for the industry this has become quite efficient. That is why now CFD has become a key component to the industrial system across the world.

(Refer Slide Time: 14:10)



So now what is essentially CFD? Or what are the key components for numerical techniques like CFD. So, if you look at it we have a problem in hand like a fluid flow problem. That is the underlined problem. And once you look at this fluid flow problem, that underlined problem we have a fluid mechanics which is one of the basic engineering system that will govern the fluid flow problem and get you to the physics of the problem. So, I have a problem in hand, I am using the concept of fluid mechanics I come down to the physics of fluid. Now once the physics of fluid is known, then I use this mathematical technique so, this mathematical technique actually will bring down to a set of PDE's.

So, Navier-Stokes equation is nothing but the set of PDE's, which will govern the continuity equations and momentum equations. So, once we get to the Navier-Stokes equations. That is where you apply your numerical methods. And get a form which we will call the discretized form. So, this is where your mathematics is applied, and applied mathematics will take you to the discretized form; that means, now the PDE's are kind of converted to a some sort of a algebraic expression. So, this is the system or the process where things become the algebraic system. Now you apply this discretized form to the geometry or in combination with the geometry which will get you to the great, and then here you use the concept of the programming language. This is where programming language come into the picture.

So, once you have the programming language into the then you get a computer program in place. Now this computer program you fit this setup essentially the program is a set of instruction for the computer to do. The set of instructions when it is sent to the computer you get a simulation results. Now, that simulation results you look at the results you do comparison you do analysis and then finally, it will represent the basic fluid flow problem. So, that actually if you look at it that closes the complete loop. So, you start a physical problem here, this is my physical problem. So, this is the physical problem to start with. So, once you start with the physical problem, through this complete process you close this loop and come back to this particular system. And this is what is CFD all about.

That means, I have a problem in hand which is a realistic problem, and I have to solve that problem through this numerical technique, and this is how it is done. It takes care of the basic physics. So, that is why the physics of fluid comes into the picture, it takes you to the mathematics so, that gives you the set of partial differential equation. Once the partial differential equations are formed so, the numerical techniques it gets to the algebraic form, then you represent your physical problem through the grid we will discuss about all these in details as we move along the courses.

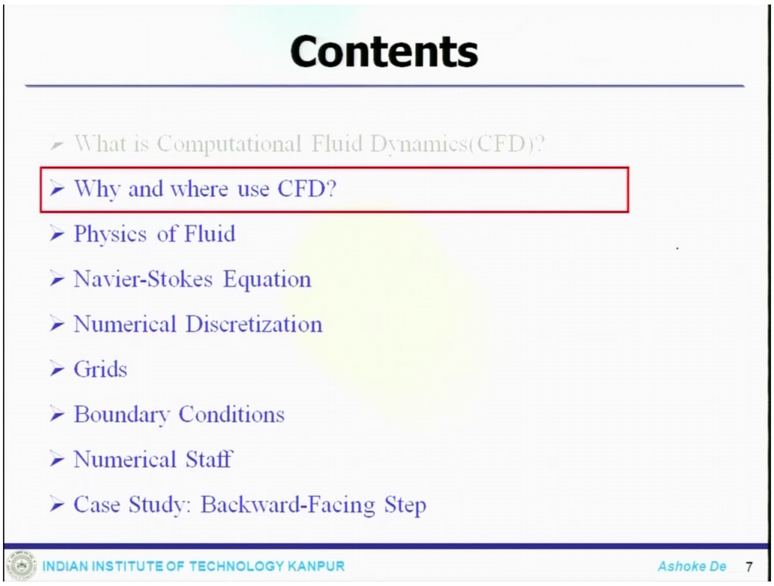
Then the set of programming code you generate through the programming language is could be FORTRAN this could be C, this could be C plus plus, any program visual C visual C plus plus anything is possible. This programming language will get you the computer program, and the computer program when you said that set of instruction you sent to the individual computer then you get the results. Now as of today the computing platform or the architecture has been show powerful that one can do a realistic situations in a quick turnaround time so that time frame of the solutions has come down because people do multi-processing system.

So, you have a thousands of course, billion of course, available today and solving, but 3 decades ago this was a challenge how to solve a Navier-Stokes equation using computer programming because the computer architecture was not powerful. But today as we have a very powerful computers and due to the availability of this powerful computers;

actually we can get this solution done in a quick turnaround time. So, this is what is all CFD is about. So, once if you look at a complete building block, that give you the complete pictures. And this has the essence of the complete course that we are going to have in this particular system.

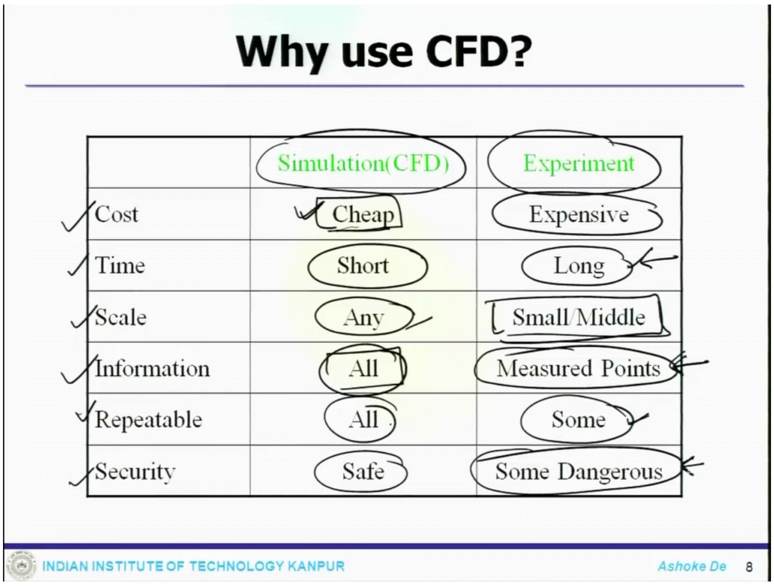
So, it will take care of your physics, it will take care of your equation, it will take care of your discretized form, it will take care of your programming, it will take care of your solution method. So, essentially CFD is all about and all these individual components we will go on discussing in details. And finally, we should be able to solve a problem in our hand that is the objective of this course.

(Refer Slide Time: 19:15)



Now, as I said why and where we use CFD.

(Refer Slide Time: 19:20)



Now, again if you look at from the comparative perspective; when you talk about CFD and when you talk about experiment. If you put them side by side this is where the CFD gets, you the simulation results and experiments which is more preferred method. But now you look at 1 by one component, if you look at the cost. As I said this is why it is more preferred or accepted in this is, because CFD becomes much more cheaper.

Ones you have a framework or computational framework ready, then the simulations of CFD solutions are much more cheap compared to experiment where experiment is much more expensive now if you look at the perspective of a industry especially the production industry who has to manufacture things now for them getting a design data is really a challenge. And now what people use to do they use to go by correlation methods from correlation method using the factor of safety they used to design the system, and go for fabrication get the testing done and then come back to the final design.

But now with the advancement of CFD and being cheap, now people get the design data through the CFD technique. And it is much more faster as I said one can have a turnaround time in a quick fashion. So, the thing as much cheaper compared to experiments so, that is one of the key component. Now, as I said time CFD solution time is much shorter compared to experiment. Because if you look at the experiments you have to first design the experiment, then you have to send it for fabrication when the fabrication takes place then you have to look at the surface finish and everything finally, you setup the experiments and then instrumentations then finally, you get the experiment done. And then you look at the data to analyze.

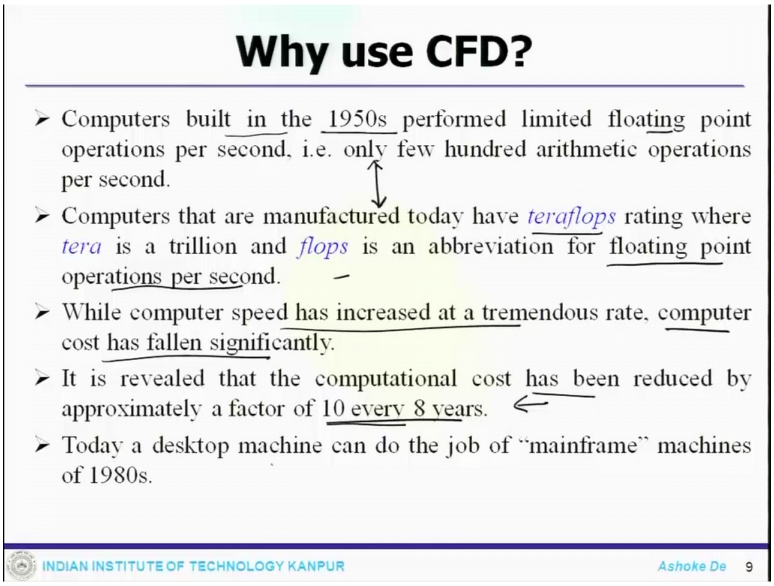
Scales if you look at the scales CFD can be applied to any scale; that means, it starting from a flow around a particular choke, this is the scale I mean even smaller than that; where we talk about microfluidics, it can go to any scale. I mean flow around a aircraft, flow around a rocket vehicle anything can be done. But experiments for a large scale problem it is not that kind of easy task. So, scaling is a big problem, but for CFD with the available computer we do not have that kind of problems we can easily solve any scale of problems. So, starting from a microfluidic channel going to the space everything can be solved through the CFD, that is the scale we talk about in CFD.

Now, information, information means when you solve the physical problem, you would like to see everything in and out of that system. And only using CFD we are able to get complete information of the CFD. But when you do the experiments compared to that you have only set of major points; that means, when I design a experiments a priory I will put certain design points or the measured points and only those locations I will have data. Apart from that I cannot have data. Big said that; that means, if I have to solve a flow in a channel, which is one of the most simple and regularly used system I can have every portion of the channel the information that I can get from CFD.

Whereas, using experiment I can get only few locations and certain planes and all, these are the restrictions. Repeatability if your mathematical tool is powerful enough; that means, your CFD technique is powerful enough your coding is good, you have a high accurate methods, if your solution technique is accurate it is stable, it is then it is always repeatable. Anytime any moment you conduct the same numerically experiments you will get the same set of results on a particular geometry, but experiments the repeatability is always a challenge. I am not saying that it is not achievable, but it is always a challenge. And the problem becomes more and more complicated this becomes more and more challenging.

Now, if you talk about security, this is much more safe in the sense. It is not nobody is going to steal your system it is like it does not harm anybody. But experiments it is always vulnerable to certain sort of hazards because, there is a human intervention which is involved. In that sense, design of experiments to the setting of the experiments along with the equipment's or the instruments that is where the danger lies. And that is why all these if you look at and put together. The comparison says we have a numerical method in hand which can be powerful enough to get you a solution done using CFD. And that is why over the last few decades 2 to 3 decades CFD has become a key component or key tool for industrial applications. Not only for the academic perspective or the research point of the show it has become a key component to any industrial applications.

(Refer Slide Time: 24:27)



Now, why use CFD is also if you look at as I have been saying. So, when you send those set of instruction to the computers the computers which are built in 1950's or 60's they had limited system of doing the floating point calculations; that means, the computing architecture was not that powerful to get you solutions in a quick fashion. But only few arithmetic operations could be done. Essentially if you look at the complete picture of CFD set of governing equations are converted to the set of algebraic system. So, essentially there is our mathematic system, and finally, those algebraic system means you need to do certain floating point calculations.

And floating point calculations in fifties 60's was very hard to get through the computer; because of the architecture they are not that powerful. Now as of today, if we talk you can do not only teraflops, we can do petaflop calculations. So, this is the advantage, if you look about these 2 era there is a huge difference. I mean 50, 60, 70's getting a simulation done was a challenge today getting a simulation done is not at all a challenge.

Because the kind of powerful computer we have we can get a teraflop petaflop kind of simulations done. And these flops mean is the number of floating point iteration per second. So, that you can imagine what kind of speed we are dealing with and that is why this has become CFD has become a key tool for the industry because, they can now afford to use it and get a solution quickly.

Now, in addition to that it is all about time. That is what we have been talking so far now if you look about the other cost like. Not only the speed has gone up over a decade the price has also gone down; that means, it is a win-win situation, our speed has gone up at the same times we are not getting hit by the price also computer price has gone down. So, that has given a ideal platform to conduct the targeted numerical experiments. These targeted numerical experiments will allow. Now if you look at the computational cost has been reduced approximately by of factor of 10 in every 8 years.

This is a number which is a approximate number there is nothing hard and fast about this number, but just to give you an idea what is the rate of this price which is coming down, I mean with the increasing technology; that means, what is the technology; that means, in the computer architecture. And that is where our computer engineers electronic engineers they are working hard day and night getting a high end computers to us. And we as a end user we use of that computer get our physical problem solved. So, that we have a solution in a quick turnaround time.

Now, even if you just take a comparison of that. Today a desktop machine can do a mainframe problem for which used to exist in 1980’s. So, that is give you an fair amount a idea what has gone from fifties 60's 70's to now as of today at this date what is the architecture has gone up, computer price has gone down, and we can do petaflop calculations or floating point calculations. So, we will stop here today. And we will take from here in the follow up lectures.

Thank you.