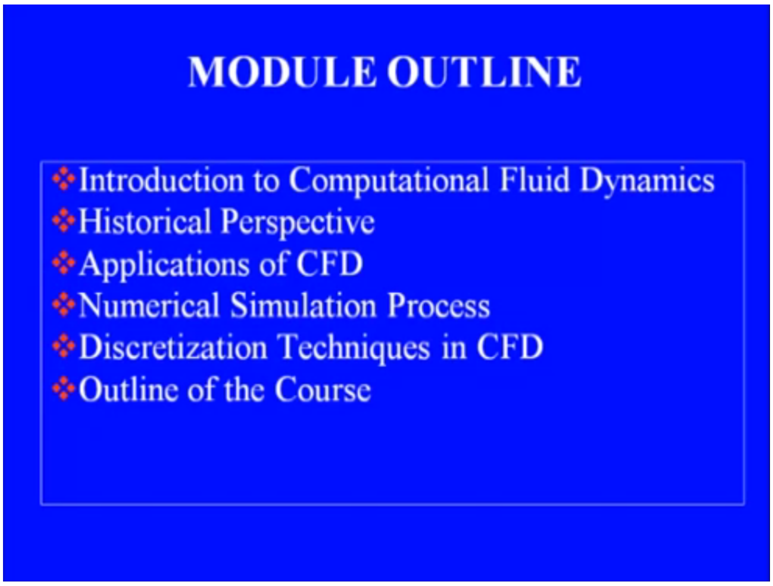
Computational Fluid Dynamics
Dr. Krishna M. Singh
Department of Mechanical and Industrial Engineering
Indian Institute of Technology - Roorkee

Lecture - 01
General Introduction: Historical Background and Spectrum of Applications

Welcome to the NPTEL course on computational fluid dynamics. I am Krishna M. Singh from Department of Mechanical and Industrial Engineering, IIT Roorkee. In this course, we will have a fabulous journey to the fascinating world of the computational fluid dynamics. You will have a look at all the aspects theoretical, numerical, and application aspects of computational fluid dynamics, which will enable you to apply it to solve myriad problems in real life.

In the first module, we will have 2 lectures wherein we will provide a brief introduction to computational fluid dynamics.

(Refer Slide Time: 01:16)

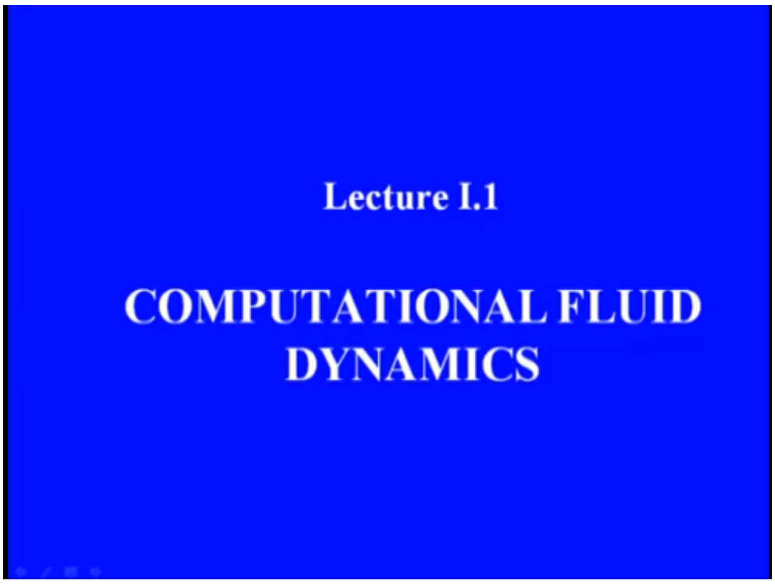


We will briefly touch upon the basics of what we call computational fluid dynamics, will have brief look at the historical perspective how this method evolved and how this particular branch of computational mechanics has evolved with a time and we would have a look at what are the various applications wherein CFD is being used in industry as well as in research.

Thereafter, we will have a very brief look at the numerical simulation process, which is at the heart of CFD and what are the main discretizing techniques, which are used in CFD? and then we will also briefly tell you the outline of this whole course of around 40 to 42 lectures. What are different topics which we are going to cover in this introductory course? This what we will have a look at in detail.

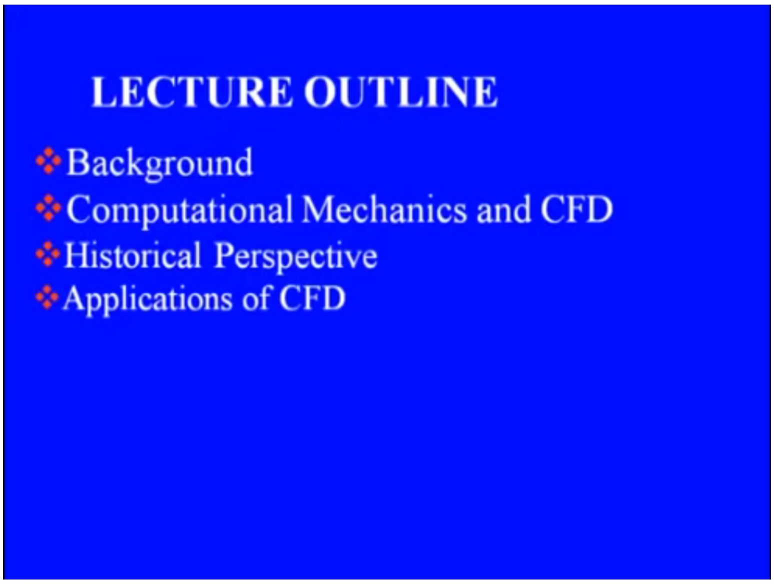
So in the first lecture, we will cover the basic introduction to computational fluid dynamics, historical perspective and applications. The remaining 3 topics we would cover in the next lecture.

(Refer Slide Time: 02:22)



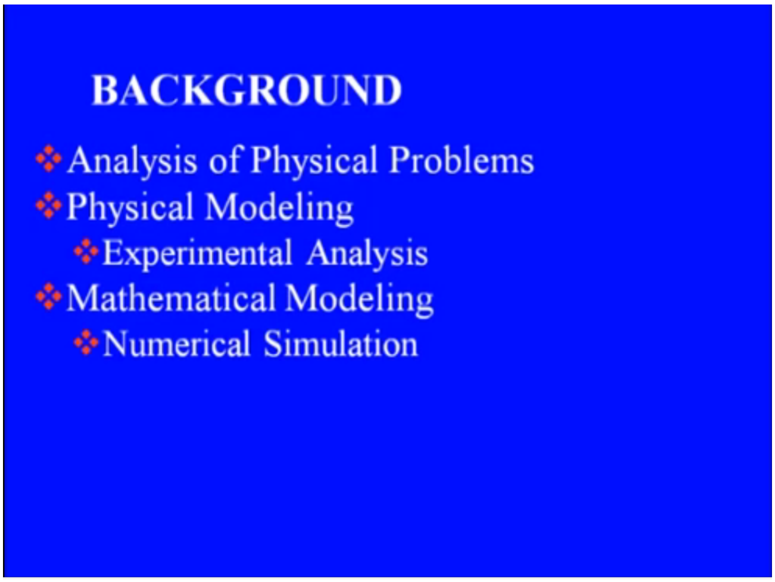
So now let us have a look at the basics of computational fluid dynamics. We will have a look at the background, how this particular branch has evolved? What is its application? Where does it stand today and what is the connection between what we call computational mechanics and CFD?

(Refer Slide Time: 02:41)



How are these 2 specialized or related to continuum mechanics? Then we will have a brief historical tour of the evolution of computational fluid dynamics and then we shall have a look at few important applications of CFD.

(Refer Slide Time: 03:09)

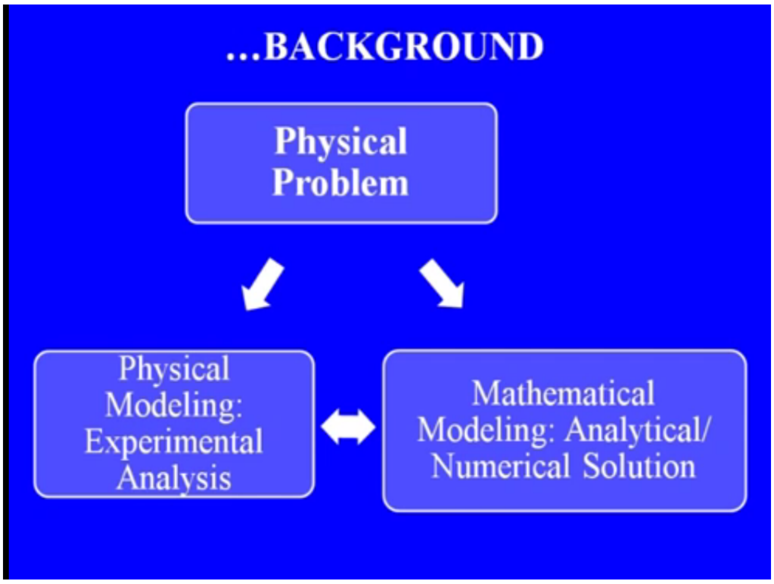


Now background of the CFD, let us have a look at what is our main objective as engineers or physicists? We want to analyze a physical problem and we can tackle physical problem in 2 ways. First way what we call physical modeling, which would involve experimental analysis wherein we will build the prototypes of a physical system, instrument it, collect data.

And analyze what is happening in different parts of the system on the different operating conditions or what we call environmental conditions. The other part the way we can analyze a given physical problem is to come up with a mathematical model, which will result in the integral or differential equation. This integral or differential equation may not be amenable in fact most of the time for real life problems. It is not amenable to what we call closed form analytical solution.

Then the recourse is that we can go for approximate numerical simulation and this particular aspect has picked up specifically in past half a century with the developments in computer hardware and computer science. So let us have a look at the entire relationship between both of these approaches to solve a physical problem.

(Refer Slide Time: 04:36)



So we are given any problem, this physical problem could be from any branch of physics or mechanics or engineering nor they are 2 approaches the first one discussed that we can come up with a model, we can fabricate a model of a physical system, instrument it and perform what we call experimental analysis, collect the data, interpret the data and thereby come with the conclusions about the physical behavior of the system.

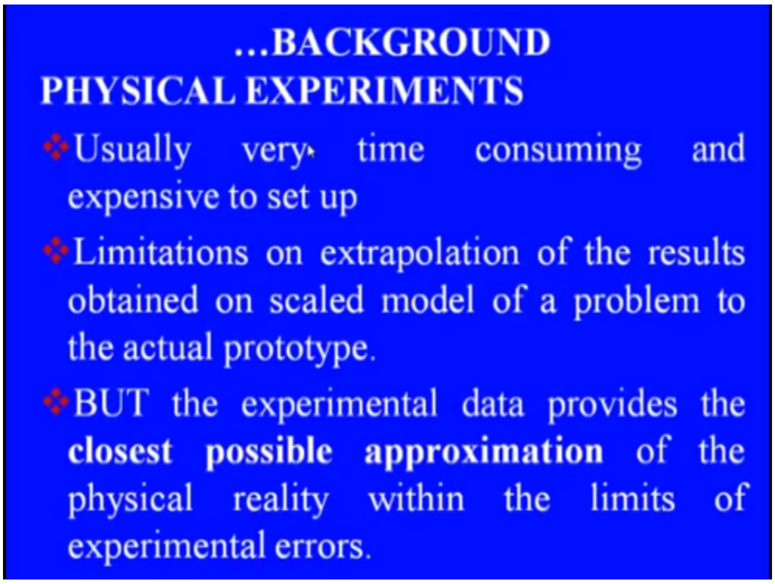
The second aspect is we make a mathematical model and try to perform analytical analysis if we can, if analytical solution is not possible, we can also have a numerical solution and the way these arrows indicate the both of these physical modeling with experimental analysis and mathematical modeling they both support each other, they do not adjust in isolation, one is not a replacement of the other.

This point I would like to emphasize to anybody physicists and engineer or an analyst at place be where we cannot do away with either of these techniques. We need both of them together. Physical modeling or experimental analysis is a must to verify or what we call validate the numerical solutions and the numerical solution could also give us indication that how we can come up with the better physical model and how we can improve upon our experiments.

So these are always used to supplement each other. So both of these approaches would supplement each other in tackling or in solving our physical problem. Now this physical problem could relate to the design, it could relate to the analysis of some problem, which we have observed in the existing system and so on. So whatever might our physical problem be we would use in real life both of these approaches in harmony.

Now let us have a look at the aspects or features of each of these 2 approaches for the analysis of a physical problem.

(Refer Slide Time: 07:01)



First about physical experiments, now physical experiments are usually very time consuming and expensive set up. We have to come up with a model, physical model must be fabricated, we have to instrument it and we have to come up with what we call fairly costly equipment to set up the measurements. The measurements must be set up in a very controlled environment so all of these aspects consume lots and lots of time.

And it has to be repeated for each system or subsystem separately and usually it is not possible for us to perform the experiments on the actual prototype. We cannot perform experiments on this an actual full size car or a train or an aeroplane. So what we normally do in the physical experiment is that we come up with what we call is scaled model, which is a much smaller geometric model of our actual system.

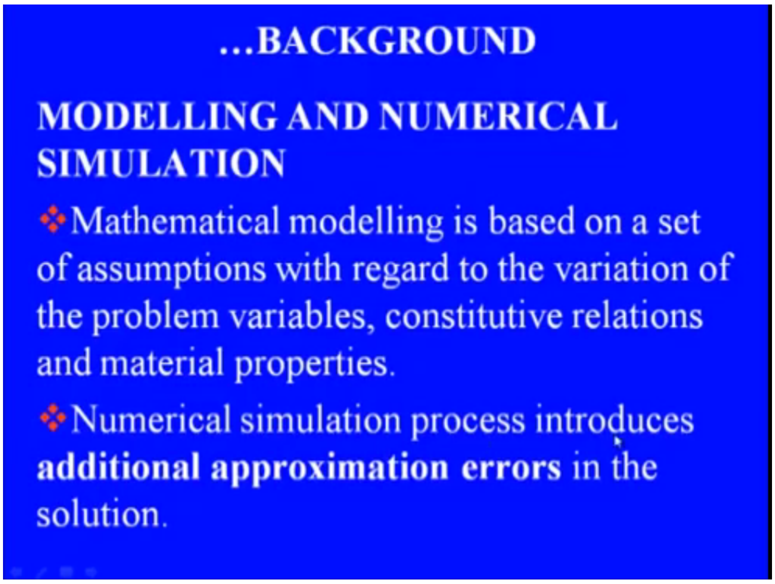
The measurements are performed on this model. We obtain certain results and there are limitations on the extrapolation of these results, which we have obtained on this scaled model to the actual prototype the both kinematic as well as dynamic constraints on this extrapolation process so this intend the limitation of physical experiments.

At the same time, the experimental data is the one, which provides us with a closest possible approximation of the physical reality within the limits of the experimental errors. We do not make any assumptions regarding the variations of material properties or what we call constitutive models or experimental simulations we would mimic the way this or actual system or prototype is supposed to work.

Similar sort of conditions we would normally mimic for experimentation on a scaled model. So that is why experimental data is very crucial and provides us with what we can call the closest possible approximation to the physics of the problem or what physical realities of the problem. Of course, we should also be aware of the limits of experimental errors. There are errors involved in each measurement process.

But not withstanding those measurement errors the experimental results provides us the best possible approximation of the real physics of the problem. So that is why we can come up with a mathematical model, which can be solved analytically or arithmetically. We have to verify or we have to validate for mathematical model with the results obtained on this scaled model through physical experiments. So physical experiments are very, very crucial in our understanding of a particular system.

(Refer Slide Time: 10:16)



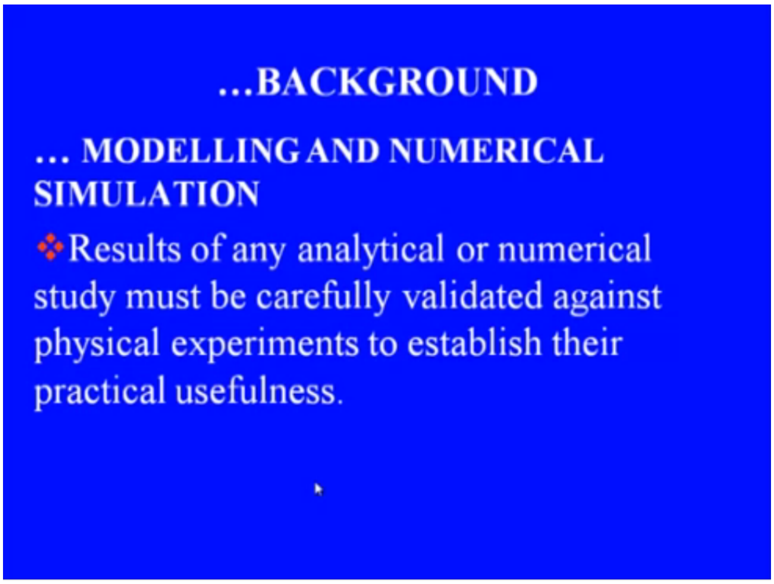
Let us have a look at in the modeling and simulation approach. Now when we come up with a mathematical model of a physical system say let us we want to find now what happens, what the flow around an aircraft body? We make a set of assumptions regarding the variation of the problem variable at which of the flow parameters (()) (10:43) of an aircraft problem are important which we need to analyze.

Constitutive relations that if say in which way different variables of the problem are related to the material properties. How do the material property themselves vary? So we make suitable assumptions regarding these aspects. So the mathematical model, which we obtained, may not be an exact representation in the physical problem. So there are approximations involved in mathematical model.

Moreover if a mathematical model cannot be solved exactly using analytical tools, we have to go for what we call numerical simulation and we will see very clearly in this course when we develop various numerical simulation scheme. We will clearly say that numerical simulation process introduces additional errors in this solution. So irrespective of the beautiful software HP might have for modeling a problem or solving a problem, which can provide us very beautiful graphics.

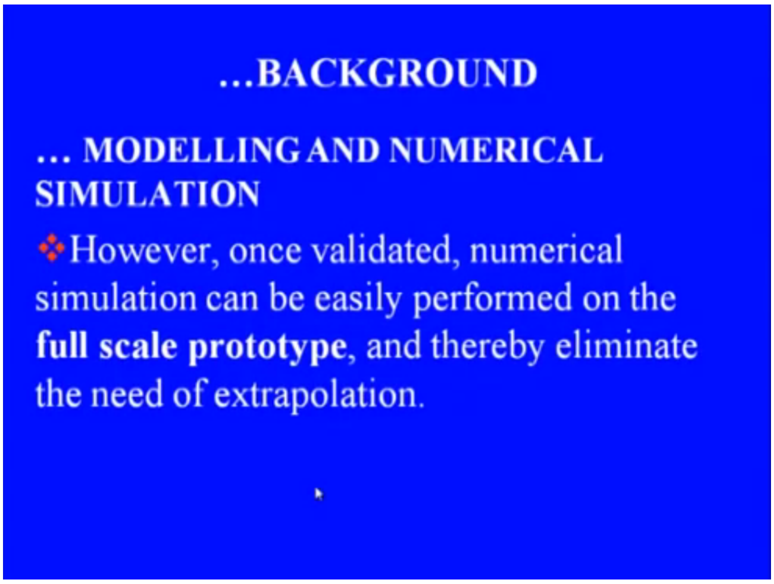
And what it can claim that if you got powerful enough computer system we get very accurate results nevertheless the mathematical model, which is basis of that software and the numerical simulation process using which we have obtained of a solution. The both involve certain approximations and as an analyst whether you are a physicist or engineer, we must be aware of the errors involved in the modeling and simulation.

(Refer Slide Time: 12:32)



So this reason why the results of any analytical or numerical study must be carefully validated against physical experiments to establish their practical usefulness. If we cannot do this we cannot extrapolate or we cannot use the results which we have obtained from the numerical simulation for a real life design applications. We will always have sudden doubts in our mind whether the assumptions, which are made they are good enough to give us a workable approximation of the physics of the problem.

(Refer Slide Time: 13:09)



With this one plus point here that once we have validated our numerical model using the results from the physical experiments, which would have usually been performed on a scaled model. So we are no more bound by the constraints of the geometry in computer simulation. There are limitations on our physical experiments that we cannot perform physical experiments on the full size prototype of an aircraft or car or train.

So we have performed experiments because of these constraints on a scaled model, use these results from scaled model to validate for numerical simulation process. We have taken the same physical model a scaled model. We will modulate in the computer what we call solved modeling. We will come with geometric model. We will use our mathematical model for this particular geometry and obtain our numerical solution.

Validate this numerical solution using the results on the scaled model obtained experimentally. So that is what would validate what numerical simulation process was that is done. We can change our geometric model in a computer and we can come up with the full scale prototype of the plain and can obtain the results on our full scale problem. So thereby you would eliminate the need of extrapolation.

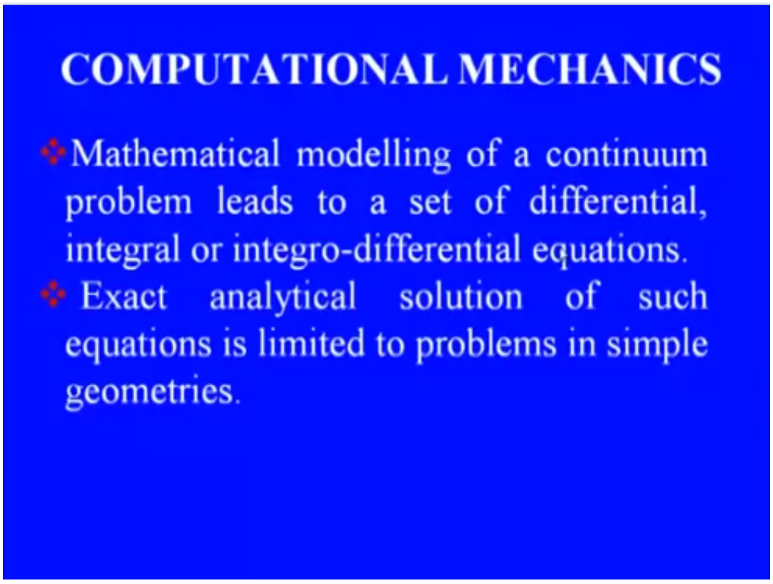
So whatever additional errors might have been incurred because of extrapolating the results on scaled model to full scale prototype that is simply eliminated using computer simulations. This is the beauty of the computer simulation not just that there are situation wherein or there are physical problems for which we cannot come up with an experimental model or physical experimental model.

It might be too dangerous or it might outright impossible for instance we have let us say we want to simulate loss of coolant accident situation in a nuclear reactor. It is very difficult to come up with a physical model of a radioactive reactor and perform this loss of coolant accident situation, but there is no such limitation in our computer model or numerical model.

They can model the full scale nuclear reactor and try to see what happens if there is loss of coolant accident situation to see how the things change, how the temperature shoot up in the reactor core and so on. So there are many such situations, which are too dangerous to come up with a physical model. They can be easily done or we can come up with a computer model and obtain insights into the working of that system in certain situations.

So that is the beauty of this advantage which we have got in the numerical simulation process and that has been one of the reasons why in past 6 decades there has been tremendous progress in the branch of mechanics, which we call computational mechanics and this has been closely associated with the advancements in the computer science both in software and hardware side and applied mathematics.

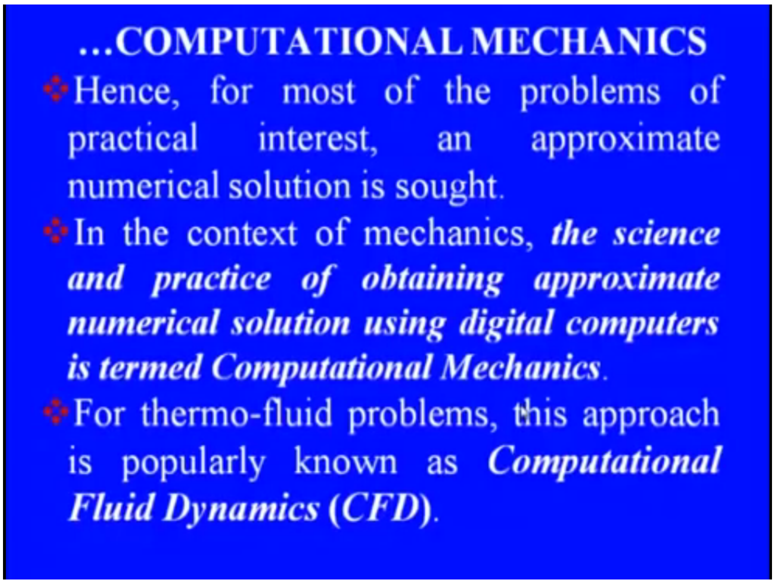
(Refer Slide Time: 16:49)



So what is computational mechanics? Mechanics is a branch of physics. What is this additional prefix computational mean here? We already know if we have got a physical problem we can come with a mathematical model and a mathematical modeling of a continuum problem it leads to a set of differential integral or integro-differential equations, which type of equation we will come up with that would depend on the mathematical model which we have used and our physics of the problem.

And exact analytical solution of such equations is limited to the problems in very simple geometrics for very simple variations in material properties. We have geometrical properties should be either constant or this would be linearly varying or so on. So there are very limited set of situations and which we can obtain an analytical solution to a problem.

(Refer Slide Time: 17:54)

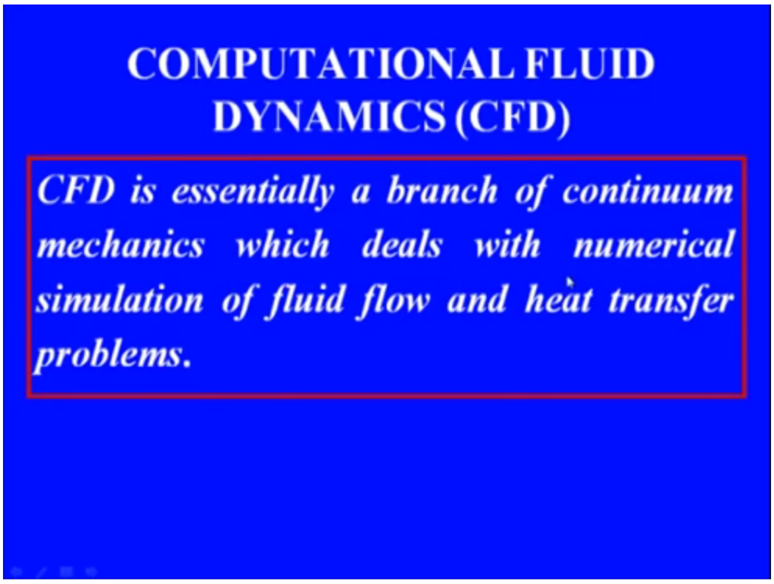


So for the most of the problems of practical interest specifically for engineering systems, an approximate numerical solution is sought and in the context of mechanics we can put it as the science and practice of obtaining approximate numerical solution using digital computers is termed computational mechanics. So remember there are 2 words here the science and practice.

So the computational mechanics is a science at a same time it is also an art form which comes up within practical experience so science and practice of obtaining approximate numerical solutions using digital computers for the problems in mechanics that is what we will call computational mechanics and in case if you are dealing with thermo-fluid problems that is problems involving fluid flow and heat transfer, this approach is termed as computational fluid dynamics.

So now let us try to put a formal definition of computational fluid dynamics. So what is computational fluid dynamics? If that is your question, let us try and answer that formally.

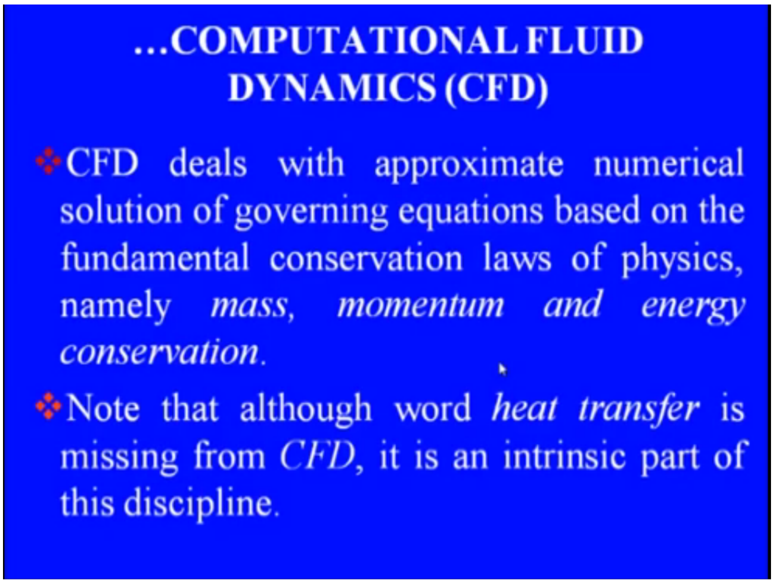
(Refer Slide Time: 19:10)



So that computational fluid dynamics is essentially a branch of continuum mechanics, which deals with numerical simulation of fluid flow and heat transfer problems. So all said and done it is a branch of physics in particular the branch of computational mechanics, but the focus is on the problem which involve fluid flow and heat transfer. So the heat transfer does not appear explicitly in the name which we have given computational fluid dynamics it is all way there, it is intrinsic part.

Heat transfer problems form an intrinsic component of the branch of computational mechanics called CFD. So let us repeat once again that computational fluid dynamics is a branch of continuum mechanics which deals with numerical simulation of fluid flow and heat transfer problems.

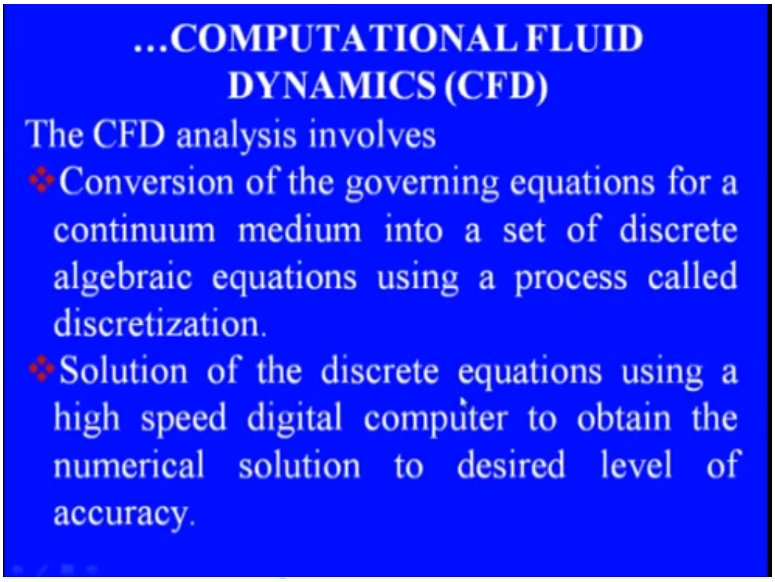
(Refer Slide Time: 20:11)



Now what we do in computational fluid dynamics? We basically try to find out approximate numerical simulation of governing equations of flow problems based on fundamental conservation laws of physics namely mass, momentum and energy conservation. So there are 3 laws based on which we would obtain the mathematical model of a problem and we would solve it approximately using digital computers to obtain the solution for problem.

In particular, we will get the distribution of the flow field, temperature distribution, density and pressure distribution for our flow system. I would once again like to emphasize that that although word heat transfer is missing from CFD, heat transfer is an intrinsic part of this discipline.

(Refer Slide Time: 21:13)



So this we can summarize what CFD involves? The CFD analysis involves conversion of governing equations for a continuum medium into a set of discrete algebraic equations using a process called discretization. The computers can only work with discrete numbers. They cannot solve our continuum problem as such, but they are very powerful to deal with what we call number crunching.

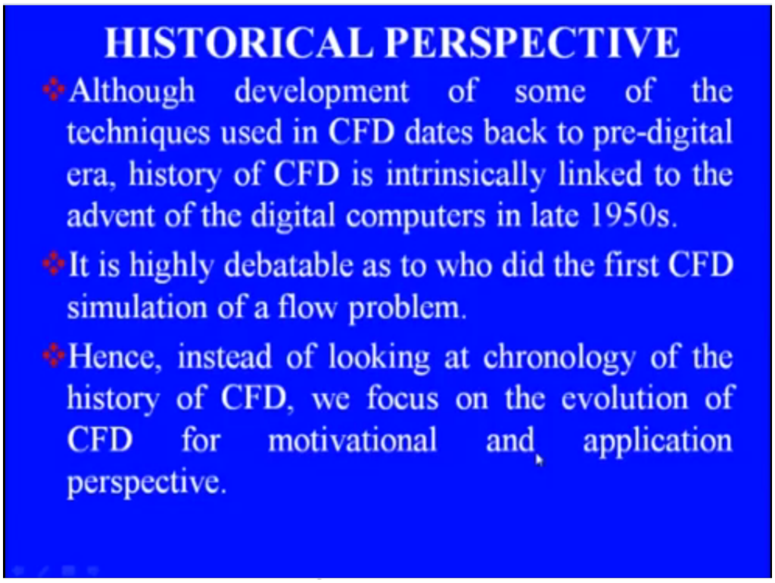
So the first process with CFD analysis would involve is to convert our continuum governing equations into a set of algebraic equations and this particular process is referred to as discretization that is to say we have converted our continuum problem into a set of discrete problems and once we have obtained the discrete problem, solve the system discrete equations using a high speed digital computer to obtain the numerical solution to desired level of accuracy.

When we are talking about desired level of accuracy, this accuracy refers to this solution of discrete equations. It is entry for those solutions over all solutions of our continuum problem per se.

(Refer Slide Time: 22:30)

Now I have defined what you mean by CFD and what in nutshell the process involves? Let us have a look the historical perspective how CFD evolved? What led to the developments in CFD?

(Refer Slide Time: 22:43)



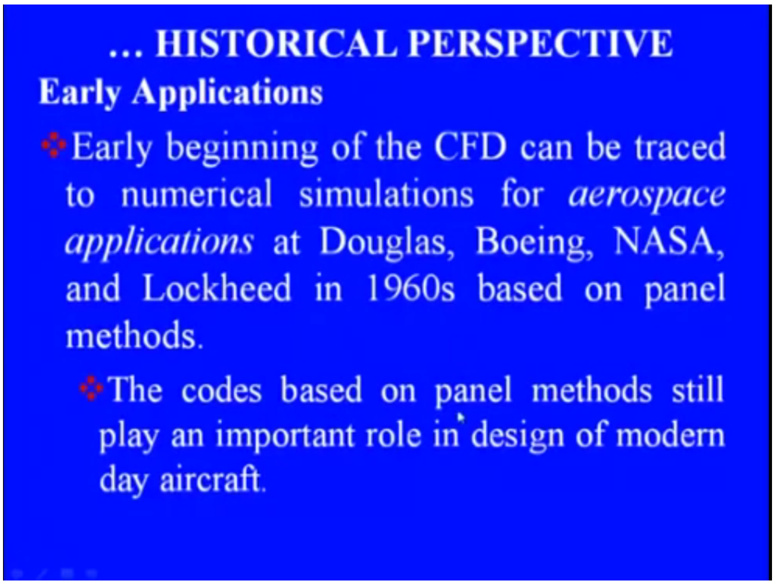
And we will see specifically when we start of in our next few modules were discussing the finite difference methods, will come across certain techniques whose development dates back to pre-digital era maybe 200 years back. So that is regarding our finite difference techniques and difference schemes. They are linked to the name of era few centuries back, but when it comes to computational fluid dynamics, the history of CFD is intrinsically linked to the advent of high speed digital computers in late 1950s.

Now this is the big debate in CFD community as to who were the first one that whenever something is invented something is established who were the first one who did the first CFD simulation of a flow problem? But we will not go into these details in our discussion today. We will not get caught into this debate. We would rather focus on the interesting developments from motivational and application perspective.

So we would not look into the chronology that is who are the first person who have performed a CFD simulation of a flow problem or who are the second person and so on. We would look at how the CFD evolved to motivate us for our course and what are the applications wherein we can apply CFD as physicists and engineers. So that is where our focus would be.

So let us have a look at the early applications and early beginning of the CFD can be traced to numerical simulations for aerospace applications. In fact, in most of the physics and engineering, the major breakthroughs or major developments would link to what we call a military superiority or developments for difference applications and CFD is no exception. So here again the first simulation were tried for the fighter planes and thereafter for the passenger planes.

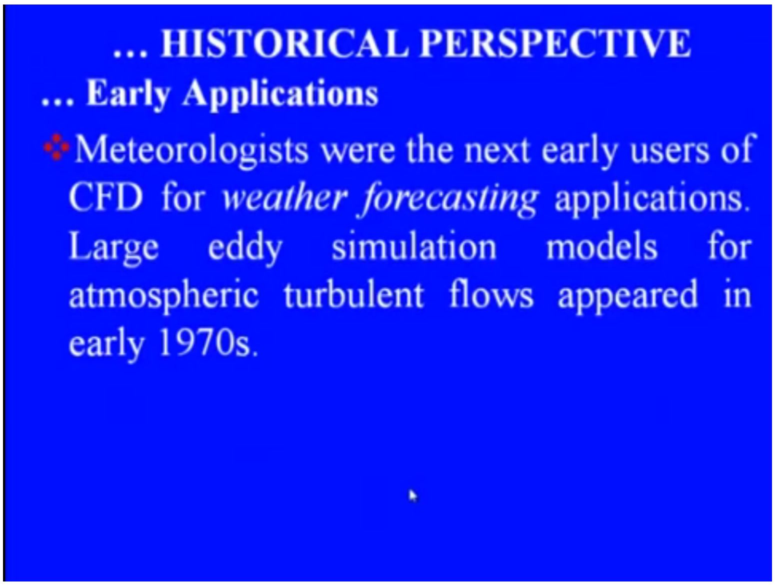
(Refer Slide Time: 24:25)



So the early beginnings were linked to the aerospace industry and developments were made at Douglas, Boeing, NASA and Lockheed Martin in 1960s based on method which is called as panel methods wherein the body of an aircraft is broken to small, small subpanels on the sudden. Distributions are assumed for some functions in terms of which we can construct our flow field.

So the panel meets with the first ones to be used for the flow simulations of an aircraft body and you will be surprised to learn that the codes, which are based on panel methods and that improvements, which were developed in late 50s and 1960s they still play an important role in design of modern day aircraft whether it is a fighter aircraft, a passenger aircraft or a freightliner.

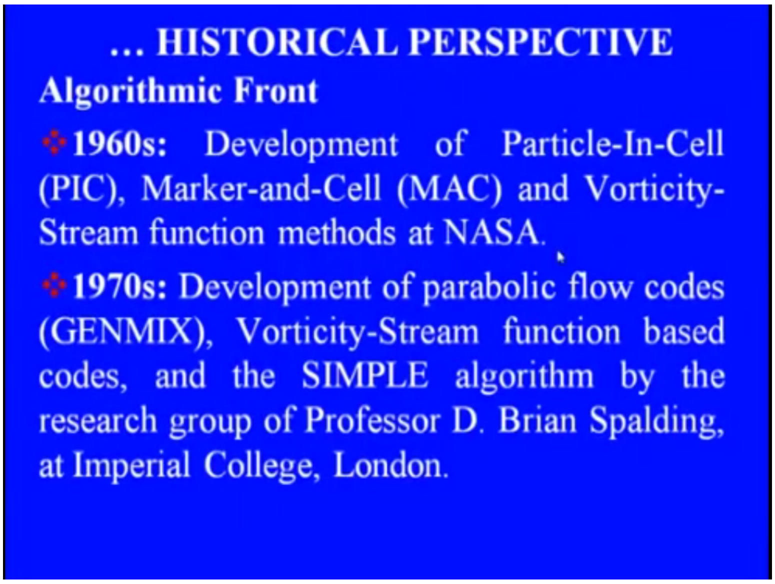
(Refer Slide Time: 26:11)



So the first design we would use for a quick analysis we would use panel methods. The next early applications were made in meteorology so meteorologists were the next user of CFD for weather forecasting applications. In fact, we will have a look at one particular methodology for simulation of turbulent flows, which is called large area simulation and large area simulation models they first appear for modeling atmospheric boundary layer.

That is to say for modeling of atmospheric turbulent flows in early 1970s. So these were 2 main industries in the early era that is in 1950s, 60 and early 70s for the developments in CFD.

(Refer Slide Time: 27:03)



Let us have a look at the algorithmic developments because just the progress in the computer hardware was not enough. We need it many algorithms to discretize of continuum problem into a track table discrete problem which can be solved using a digital computer. So there are many algorithm developments in CFD front. So in 1960 saw the development of Particle-In-Cell method and Marker-and-Cell method and Vorticity-Stream function methods at NASA labs.

In 1970s, these developments spread further and there was one school or research group led by Brian Spalding at Imperial College, London. They came up with the development of what we call parabolic flow codes, which were named as GENMIX. This happened in early 1970s, Vorticity-Stream function based codes were also dealt at Chris Sparling script and simple algorithm.

So these 3 developments, which were done by professor Brian Spalding and his student most notably professor S.V. Patankar for the architect of simple algorithm that can be said to be a watershed event in the development of the CFD because that led to the widespread of applications of the use of CFD techniques for solution of Navier-Stokes equation that is full scale flow and heat transfer problems.

Because earlier methods which were used in aerospace industry, the panel methods they were based primarily in Euler's equation and in contrast to that Sparling script provided the first thrust towards the developments in viscous flow and heat transfer problems.

(Refer Slide Time: 28:50)

And in 1980s onwards the vigorous research activity in various parts of the globe addressing different tasks of CFD. By now CFD was recognized to be almost at power with what was referred to as finite element analysis in structural applications and in fact the developments in finite element analysis computed at a design and grid generation specifically structural grid generation led to the biggest development in computational fluid dynamics as well.

So since then the research work has been going on all parts of globe. It is not localized just in Europe or USA. It is in every nook and corner of the globe vigorous research activity addressing different aspect of CFD notably development of new discretization methods, new models to deal with the turbulent flow situations, numerical algorithms, which can give more accurate and faster solution to a discrete systems.

And this philosophy robust methods, grid generation methods, which are specifically important for 3-dimensional problems in complex geometries and from practical perspective we also need to visualize the results from the CFD analysis. The CFD code gives us the flow variables at millions of discrete points. This having those numerical values tends to give us a complete understanding of our system.

Suppose processing tools which have been developed recently and visualization tools they help us in understanding the physics of the problem using our numerical simulation. In last 2 or 3 decades that has been intend the thrust that is on parallel implementation especially when the parallel super computers have become common place till 1980s this parallel computers were the pleasure only of very, very large research labs in USA or Europe or Japan, but recently they have become more widely available.

So it has been lots of research going on regarding implementation of the CFD algorithms on these parallel machines. Most recent advent in this area as the use of graphical processing units, which we have been using for playing games long since may past few decades. Can we use those GPUs, which have got tremendous processing power, which they use in 3-dimensional games? Can the same power be brought to use in the simulation of flow problems?

(Refer Slide Time: 31:57)

And from an engineering perspective, let us have a look at the way the things came into being regarding computer-aided design, which led to the industrial applications of CFD. So industrial applications of CFD have been boosted by availability of commercial CFD codes in 1980s and their integration with computer-aided design and stress analysis course. Most notably the ones based in finite element analysis.

So developments in computer-aided design and finite element analysis they are inspired developments of commercial CFD course with user-friendly graphical user interface and most of these commercial CFD course they provide their own in-built solid modeling software and very powerful visual post-processing capabilities. They also offer full interoperability with various CAD systems.

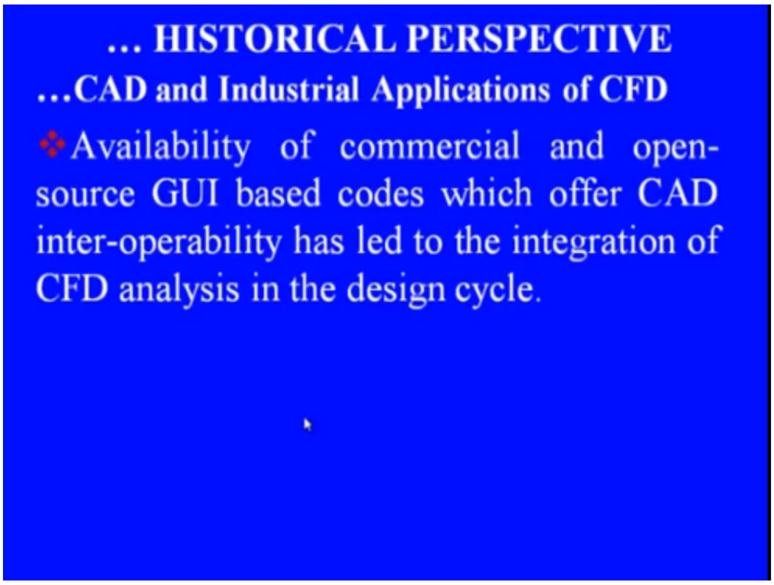
That you might have used Catia, Pro-Engineer or solid works to come up with let us say geometric model of our aircraft or a high speed train over which you want to analyze your flow health. Now those models which you might have spent considerable amount of time in developing on your favorite CAD system can be transferred directly to your commercial CFD software.

Wherein you can perform the further modification steps may be there might some geometry patching need to be done to make the geometry more amenable for flow analysis. We will perform grid generation and thereafter your CFD software can be used to come up with the flow solution. Once the flow solution is available we can analyze the velocity field, the pressure field, the track, which is being generated.

And how you can modify the geometry to reduce and track specifically say from transport applications that data can be fed back to our CAD model we can change our geometry and change geometry once we have satisfied in our design cycle that we have got or we have come up with geometry of a transport model with this plane, car or high speed train. So you now come up with a fairly what we can say final design, which meets our requirements that can be directly transferred for production software of CAM systems.

So in this way recently there has been in industry attempts been made to integrate CAD systems, finite element analysis systems, which look at stress analysis and the CFD software in 1 big integrate applications to enable the completion of industrial design cycle and thereby obtain designs of products rather quickly and without making any errors due to transfer of data from one system to another.

(Refer Slide Time: 35:20)



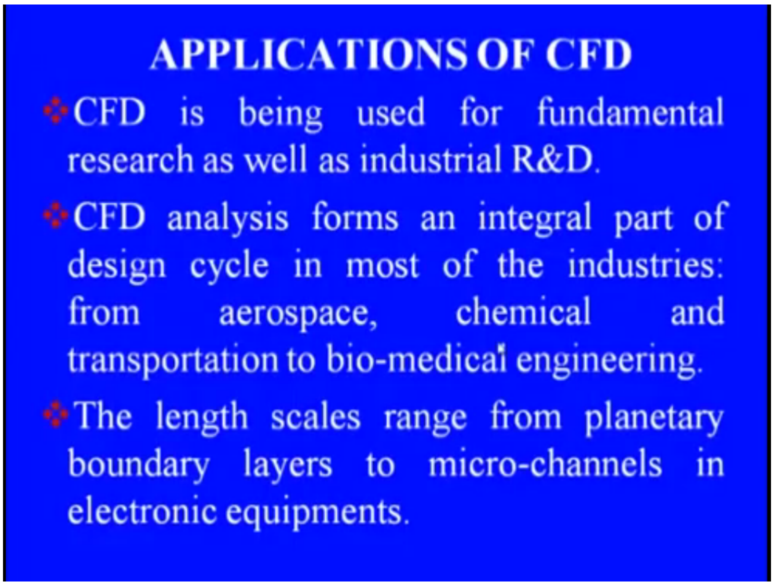
And availability of commercial as well as open source GUI based codes, which offer CAD interoperability has now led to what out say the complete integration of CFD in design cycle. Next let us have a look at the applications of CFD. Why do we need to learn this particular branch of mechanics? What are its uses?

So if you look at its potential applications, I have just mentioned few of them in our historical perspective section, but now just have a look at few other applications which can motivate us to learn this art and science of very interesting, exhilarating and demanding branch of the fluid dynamics called CFD.

(Refer Slide Time: 36:11)

And let us note it here that CFD is being used for both fundamental research, fundamental research has been carried out to understand the fluid physics in various systems at various lengths of scales starting from the planetary to intergalactic, from intergalactic to planetary to what we call micro scales.

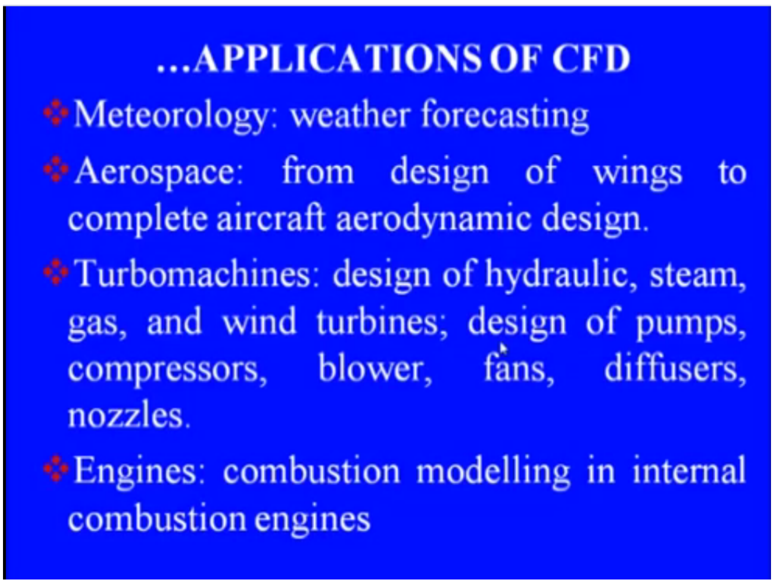
(Refer Slide Time: 36:38)



So that is fundamental research aspect. Similarly CFD is now being used thoroughly in industrial R&D cycle for both design applications as well as for what we call troubleshooting aspects. So CFD analysis forms an integral part of design cycle in most of the industries now from aerospace to chemical to transportation to bio-medical engineering and this one is the most exciting areas from the human perspective is bio-medical engineering.

And the length scales adjustments earlier they can range from the planetary boundary layers to micro-channels in electronic equipments.

(Refer Slide Time: 37:23)



Meteorologist were the early users of CFD software and they were the ones who contributed to the development of what we say the science and out of the CFD today and every day you will see on the television screens or in net cast various beautiful graphical pictures or weather forecast. Now how these forecasts are made? And the backbone is CFD software and with a set of sources what we call CFD systems.

And this weather forecasting is one area wherein the physical modeling and numerical simulation they are used together. We have got weather balloons and we have got weather stations, which keep feeding the experimental information to our forecasting stations and the other set of information is fed as initial or boundary conditions or initial and boundary conditions to our numerical simulation systems to obtain short term as well as long term weather forecast.

And this for the reasons why you will see that forecast keeps changing let say the forecast which are there in morning for some area if we look at the forecast in same agency in the afternoon there might have been a change the reason is very simple. New data was fed in form of physical measuring stations or what we call weather stations into our numerical modeling software.

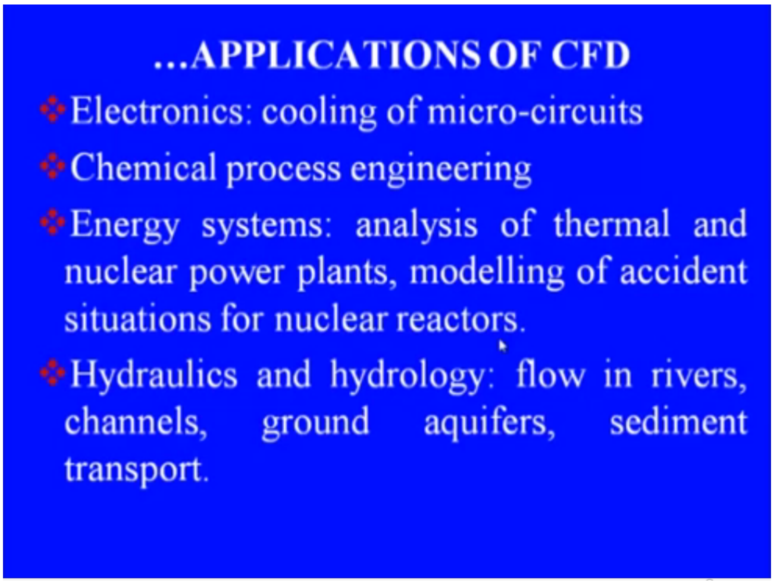
The CFD software took cognizance of the data and gave us what we can sigh a more accurate near future forecast for the weather. So this is one of the most exciting application area of meteorology of CFD, which is only the forecasting. Aerospace industries I just mentioned we saw in our historical section as for the first one to use the computational fluid dynamics is also one of the biggest (()) (39:37) today.

And CFD is used from design of wings to the complete aircraft aerodynamic design so design of all the subsystems of an aircraft CFD is used thoroughly. It could be wings, it could be body or it could be the turbofan engines, which power an aircraft and same whole squad with the space craft designs as well. Turbomachines, we have got variety of turbomachines. They could be hydraulic, steam, gas or wind turbines.

We can also have what pumps of various types and what compressors, blowers, fans, diffuser, and nozzles all of these machines wherein the flow is very important. CFD is being used to analyze these systems to come up with the better design and to troubleshoot if there are any problems in this turbomachines. Similarly CFD is also being used extensively in the design of engines, it could be internal combustion engine, they could be gas turbine engines.

So for combustion modeling as well as for the flow through a turbomachine all this complete aspects we have numerical simulation models available. For combustion modeling, we have to supplement the fluid physics from the chemical kinetics of the combustion process. So this intend the exciting research areas of CFD application.

(Refer Slide Time: 41:20)



Design of electronic systems, every day the chip makers are coming up with the smaller and smaller micro-circuits or integrated circuits with huge billions and billions of processing units, which are built in to very small area and cooling of these circuits is one of the major concerns in electronics industry. So can we come up with a numerical model to understand the process of heat generation the way it is diffused?

It is being diffused across the circuit and what could the possible cooling solutions be how well a particular cooling approach would work so on. So if before we actually design these circuits and the cooling systems, numerical simulation can be performed to understand which particular approach would work the best and will keep our integrated circuits in good health. Chemical process engineering is using the CFD modeling extensively to model all types of mixing processes and so on.

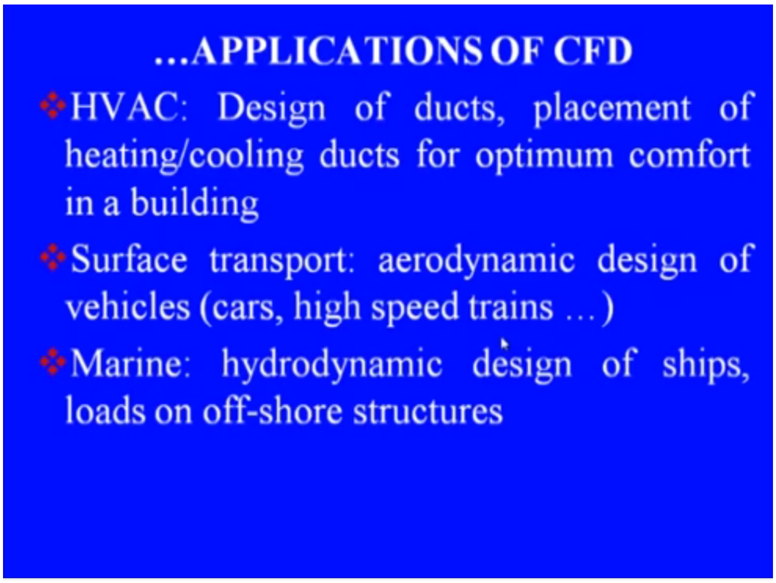
CFD is also being used for the analysis of variety of engineering systems. They can arrange from the thermal power plants to nuclear power plants to the wind farms to solar systems and so on. They can also perform the modeling of excellent situations for nuclear reactors using CFD, which are very difficult to do otherwise using physical experiments to understand what will happen in case of a unforeseen accident.

So CFD analysis can be used to (()) (43:05) that what would be the situations which would arise in excellent situations and what sort of measures should be taken of course we built-in in the design of a nuclear reactors. Understanding of the fluid physics in nature is also one of the most exciting application areas of CFD specifically for civil engineers and hydrologists. They would like to have a look at the way the flow takes place in the rivers.

How do the rivers change their course during time? How do various canals and channels they work with how the flow works there? If we have got bridge piers, how the flow is affected by the bridge piers or how the bridge piers will be designed to have the minimal effect on the flow? How the flow patterns affect what we call local environment ecology because these are natural systems like rivers, channels, ponds.

They also have a thriving ecosystem and the understanding of the flow pattern is also important to understand the ecosystem and maintains balance. Similarly, how the flow takes place in ground at first? How the sediment is transported across a river or a canal? These are all very important aspect from the civil and ecological perspective and in these areas CFD is being used extensively to support the physical experimental modeling.

(Refer Slide Time: 45:04)

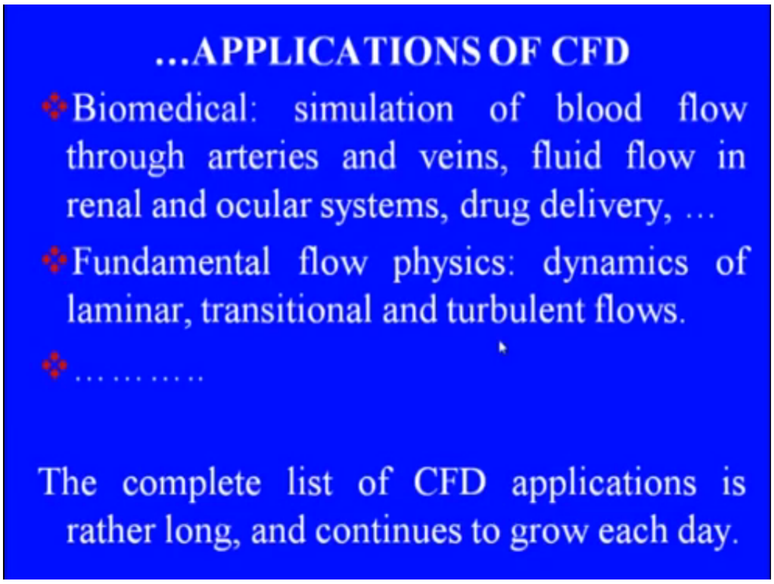


Intend the area is relevant to what we call heating ventilation and air conditioning for very large commercial hubs or maybe less if hospitals for labs, we have to come up with a proper design of ducts that where to ducts be placed? Where we should have the placement of heating ducts or cooling ducts for optimum comfort level in our building? So we can predict or we can come up with alternate designs of HVAC system before actual installing our heating ventilation and air conditioning system.

So that we can have optimum cooling and heating provided in our building. Surface transport intend the heavy user we have already seen this in aerospace industry. The same techniques are also being used in the design of cars and high speed trains what we call aerodynamic design of vehicles. This is also being used in design of the marine systems, hydrodynamic design of ships.

We can also use CFD to work on the loads and offshore structures for instance we have got oil exploration trucks, which are anchored in the sea. There also we can use a numerical simulation to predict the loads on these offshore structures and thereby come up with a robust design of these structures and one of the most exciting applications in the past decades or so is application of CFD for biomedical problems.

(Refer Slide Time: 46:54)



The prominent applications have been made in simulation of blood flow through arteries and veins, fluid flow in renal and ocular system and drug delivery aspects. In fact, the recent trend is to perform what a doctor say a patient specific simulation. Before a doctor wants to perform let us say cardiac surgery or a bypass surgery for a patient, he or she would like to know that how the blood flow in obstructed mode or in diseased condition in the patient.

So we can have CT scan of the patient body. Imaging CT scan and synthesizing systems we can come up with a geometric model of the complete cardiovascular system. That cardiovascular model which we have obtained can be simulated using a CFD software to see which are the areas have obstruction? Where the doctor needs to focus on? Where will be the ideal placement for the bypass and so on?

Or in case if just the operation has been performed to remove sudden obstruction where those obstructions are and what could be the best way to tackle them? So this patient specific simulation can help a doctor to plan the operation for a patient very meticulously. Similarly, the drug designers can also understand that when a particular drug is being fed to a patient how effective would be.

The drug could be fed let us say using a capsule or a tablet or an injection form, which mode works in what way, how the chemical drug is being transport by blood to the place in the body where the targeted place in the body. So we can just have much better understanding of a drug delivery system. We can come up with the better design for the stents. What sort be the design of a stents for a particular patient even.

The correct CT data of a patient that data could be used to design good stents for the particular patient. So there are tremendous research work which is going on in this biomedical area. It is also one of the most complicated ones because here one has to collaborate between the people from diverse background. We have to collaborate with the doctors, with the patients, with the people from the animation industry who can give us a synthesized origin from the CT scans.

We need computer graphics experts, we need the experts in flow simulations, we also need experts in what we call biomechanics to help us understand a particular situation or understand what is happening in the cardiovascular system of particular patient and so on. So this biomedical applications is one of the most exciting research areas today with applications of CFD.

And how does would like to put in last fundamental flow physics? We would like to understand the dynamics of laminar, transitional or turbulent flows in different geometries and to understand this flow physics numerical simulation can play a very vital role specifically with regard to turbulent and transitional flows wherein the planning of experiments for complex geometries might be pretty difficult.

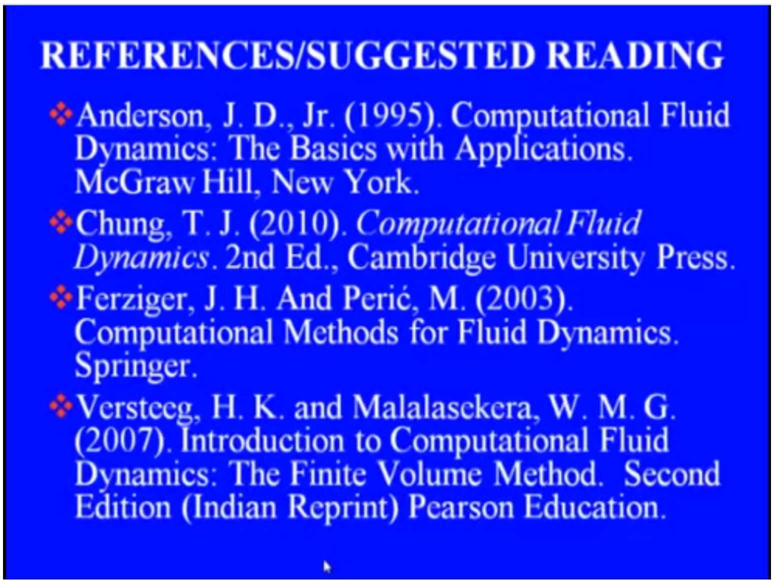
So if we can perform very fine scale or fine resolution direct numerical simulations they can give us very good understanding of the physics of the flow in a particular situation. So I just listed few very prominent application areas, which I hope would motivate you to learn science and art of CFD, which you can use for your industrial problem or if you planning to have research care in CFD.

You can already see that you have got many exciting and challenging problems in all branches of physics, engineering and biology wherein you can apply your knowledge of CFD to contribute to the design of a newer systems to design a better health monitoring systems or better you can provide health to the doctors to come with the better design of the lifesaving equipments.

You can also help them with their operations and so on. So this complete list of CFD applications is rather long, is growing each day okay. The once you get involved with your research work in CFD you will come across many, many such exciting applications. Each day you will come with one or many exciting problems. At the same time, I would like to (()) (52:25) on you the place remember is CFD also a pretty demanding subject.

It demands a thorough understanding of physics, mathematics and at least good knowledge of one computer programming language apart from a general knowledge of information technology. Now I just briefly have a look at the references which you might be interested in. I just list you a few books which if you are interested you can have a look at them.

(Refer Slide Time: 53:03)



The first one is computational fluid dynamics, The Basics with Applications by J.D. Anderson, this is one of the ways the title says basics with applications. This is one of the most interesting and simple books on computational fluid dynamics. On this extreme, we have got computational fluid dynamics by Chung T.J. which provides a compendium of the current knowledge in computational fluid dynamics.

And intend of text book you can find with that of Ferziger and Peric and we will later on discuss one particular methodology for discretizing called finite volume method. This text book of Versteeg and Malalasekara would be particularly useful. In the next lecture, we will also come across few more references. I will put across few more books which you might like to consult.

So we will stop here today. In the next lecture we will have a look at few more introductory aspects of computational fluid dynamics.