Computational Fluid Dynamics

Prof. Dr. Suman Chakraborty
Department of Mechanical Engineering
Indian Institute of Technology, Kharagpur

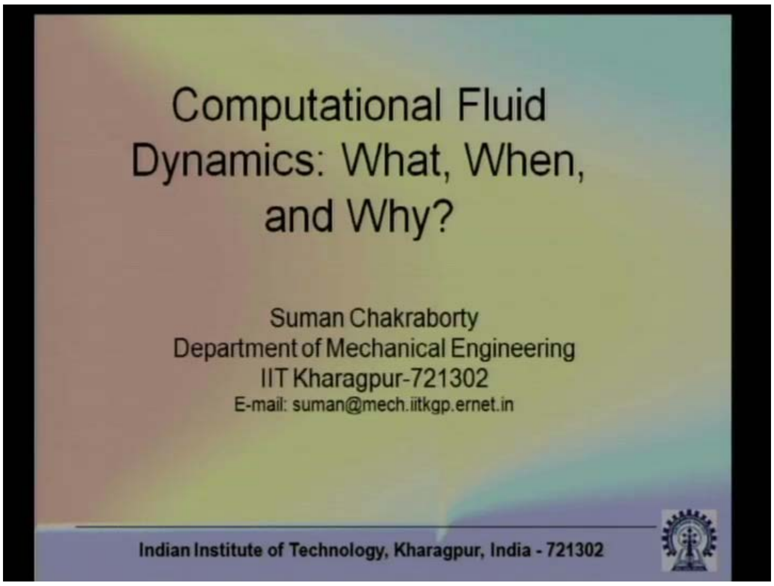
Lecture No. # 01

Introduction to Computational Fluid Dynamics and Principles of Conservation

Let me welcome you all to this introductory course on CFD or Computational Fluid Dynamics. So, in this particular course we will emphasize on the fundamental principles that govern the implementation of CFD in practical applications. Now, the first question should come that, what is CFD, why are we doing CFD and when should we do CFD. We will try to answer these one by one.

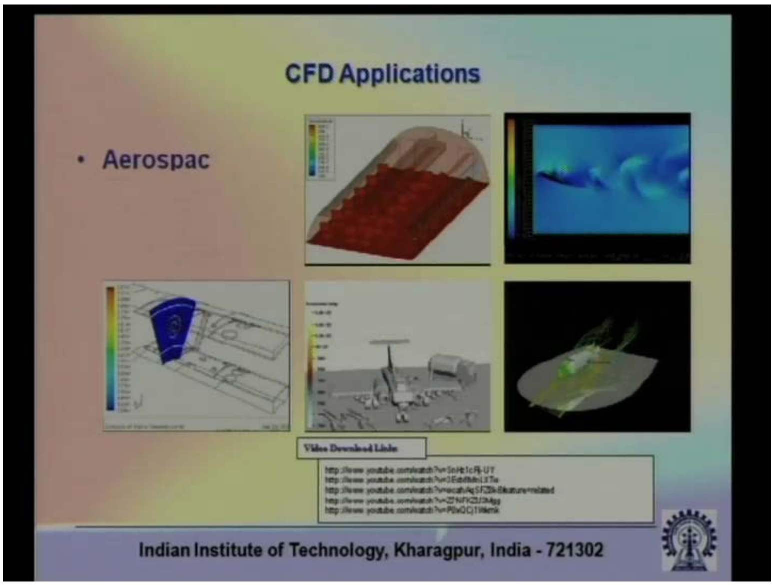
So, what is CFD? CFD has a full form of Computational Fluid Dynamics, but it's scope is not only limited to fluid dynamics. So, when we say computational fluid dynamics, we essentially mean computational transport phenomena, so which involve computational fluid dynamics, heat transfer, mass transfer or any process which involves transport phenomena with it. We will come into that in more details subsequently.

(Refer Slide Time: 01:24)



Now, next we will try to see that, when CFD; that is, what are the important and challenging applications that we have for CFD these days. So let us look into some of the interesting applications of CFD.

(Refer Slide Time: 01:32)

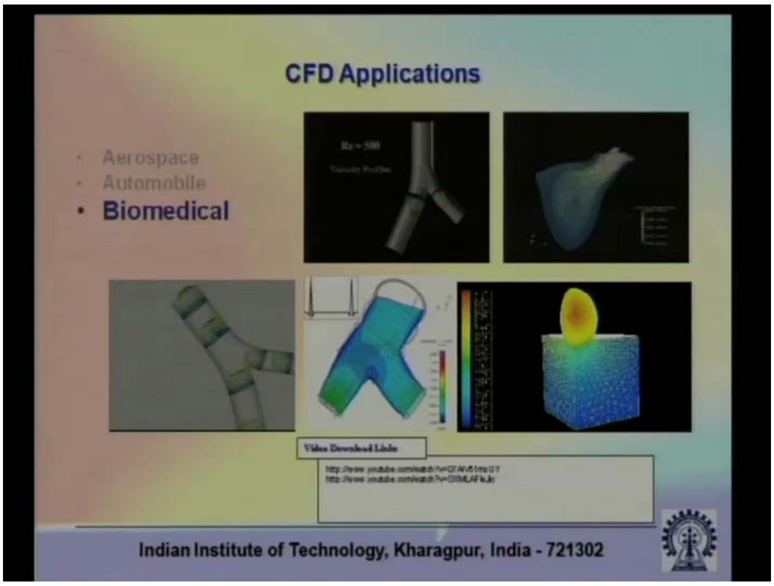


So, first application which has been the most common one is the aerospace application. So, if you see the aerospace applications, we will see that, let us consider an aircraft as an example. So, if you consider an aircraft, it has interior and an exterior. So, when we consider the design of an aircraft, you have to design the interior that is the air conditioning and ventilation system, how should it work, so there is an interior and there is also an exterior. And when you have an exterior, you are basically considering an aero file section and there is a flow past an aero file section, which you are intending to model. Also you have combustion chamber within the aircraft and you need to know about the transport phenomena within the combustion chamber. This is an example of a case when there is some unintentional incidence that is a fire that is set up in the aircraft, then what happens. And one can have CFD for spacecraft applications also. So, that is like for aerospace applications.

(Refer Slide Time: 03:00)

Next automobile applications. So, if you look into automobile applications, what you see is that again the basic philosophy is very similar, you want to see the fluid flow behaviour inside the car, as an example. The whole idea is to make a good design, so that the passenger feels comfortable inside the car, but the other important consideration is the external flow, that is how fluid flow takes place across and outside the car. Because that sort of determines the drag force on the car and that is where most of the clever design goes into. But one should not undermine the importance of design that goes for designing the ventilation system inside the car. Then there is the combustion chamber and combustion chamber design is also very important from the transport phenomena view point.

(Refer Slide Time: 04:00)



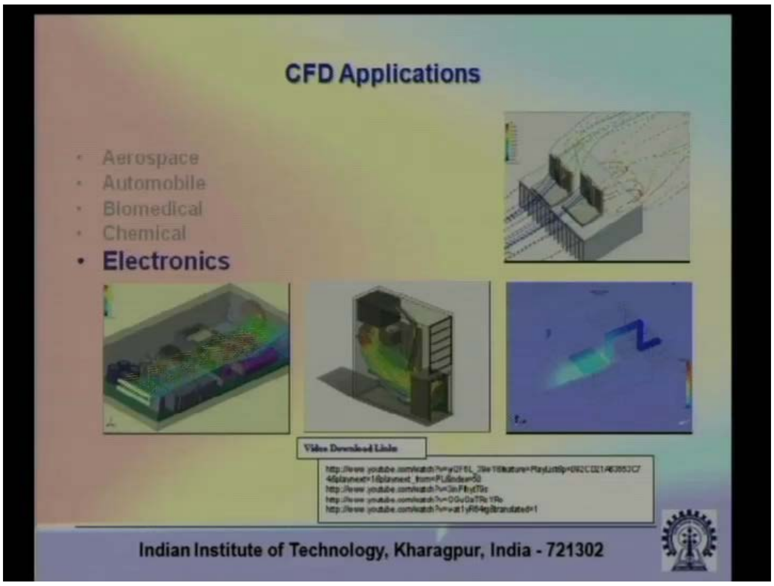
When we say automobiles and maybe aircraft applications, these are very traditional applications of CFD, and that is why we have begun with those applications. But there are many applications which are not so traditional, but which are very importantly used in modern day applications, like the biomedical applications. So, you can see that you have basically interesting flow patterns in bronchial tubes and these are some of the examples which show that how these flow patterns are there. Like this is the human heart being modeled as a sort of a pump in CFD applications. Now, this is flow through arteries and this is also an interesting case where fluid flow analysis is very important. This is an example, where it is not fluid flow, but heat transfer. So, this example is like you have a tumor and somebody is trying to have a laser treatment to destroy that tumor. So, how the temperature is distributed across the tumor on the laser treatment, that visualization is there in this view graph.

So, like for a medical practitioner this is very important because medical practitioner can get an (( )) idea that what will be the extent of destruction of the cells with the laser treatment. So, accordingly the parameters of the laser treatment may be sort of designed, so that healthy cells will not be damaged. Of course, there may be some damage, but it will not be damaged beyond a critical extent and only the tumor cells which are designed to be destroyed will be destroyed by the laser treatment.

(Refer Slide Time: 05:55)

CFD has lots of applications in chemical industries. And if you see, this is a nice example where you have mixing of two different chemical streams and the interface emits some bubbles. And so it is a very complicated interfacial phenomenon and CFD sort of tries to capture these type of phenomena by involved computational modeling. You can also have like injection of streams or separation mixing, these are all important chemical engineering applications and CFD has a big role to play in many of these applications.

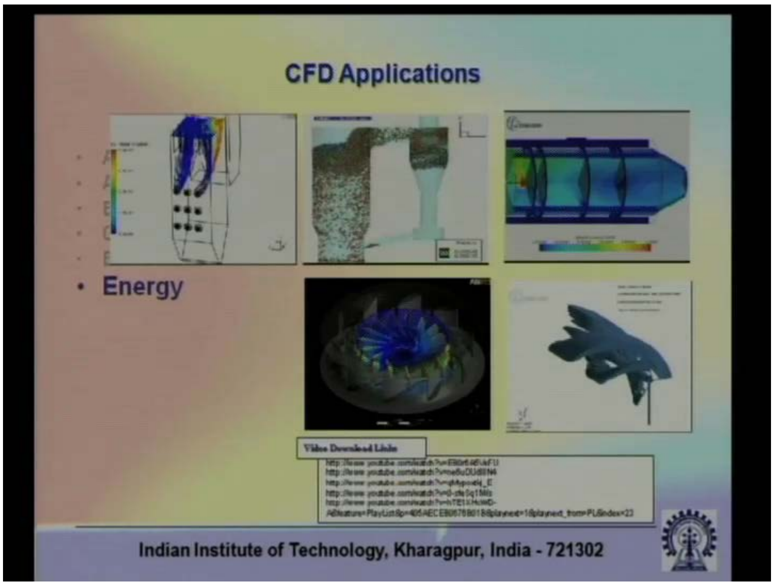
(Refer Slide Time: 06:37)



Now, chemical engineering application is a sort of a traditional application, but you can also have not so traditional applications, just like the biomedical we gave one example, another example is electronics. So, these days, like with the advent of new generation electronic devices, the sizes of electronic devices are getting more and more reduced. But if you are still having the same power rating, then what happens? Then basically what you have to do is, you have to dissipate a much greater amount of heat per unit volume than what you had to dissipate for larger devices; that means, you must have more efficient electronics cooling strategies, so like no matter whether it is your laptop or desktop or any other electronic device you require efficient cooling strategies.

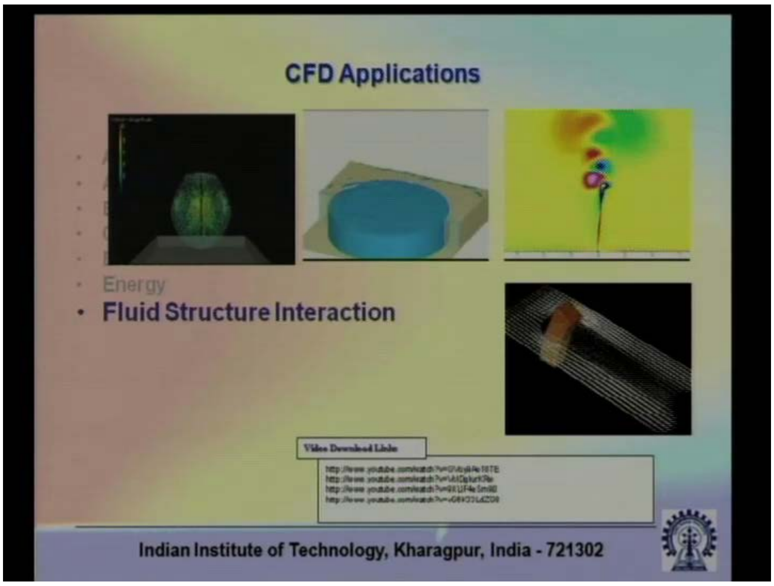
So, if you are cooling it by different means for example, if you have a fan cooling or like cooling through different other technologies, we will not go into such technologies, but there are interesting technologies these days, one has to assess that what is the fluid flow and heat transfer around the flow passages in electronic devices. I mean, it may not always be a forced convection type of cooling, you can also have a cooling because of a phase change in a loop system, which is called as a heat pipe. So, you can have for example, a system where you have a hot spot in the device and there is a fluid which gets evaporated by taking heat. And then it the fluid gets transported to a different place which is colder and then it gets condensed. So, you have a cyclic evaporation and condensation process. In this way, the entire energy is sort of handled in the form of latent heat. So, there is no sensible change in temperature and there is a sort of a cyclic phenomenon that goes on occurring.

(Refer Slide Time: 08:35)



There are lots of other energy related applications, like you can see this is an example of emission characteristics from a coal fired boiler. You can have interesting flow in fluidized based systems and separators and so on, and like, these are many different energy related applications, like for example, this is like a wind turbine. So, you can have very interesting energy related applications, and fluid flow and heat transfer and mass transfer in these many of these applications stands out to be very critical and CFD plays a big role towards understanding that.

(Refer Slide Time: 09:16)



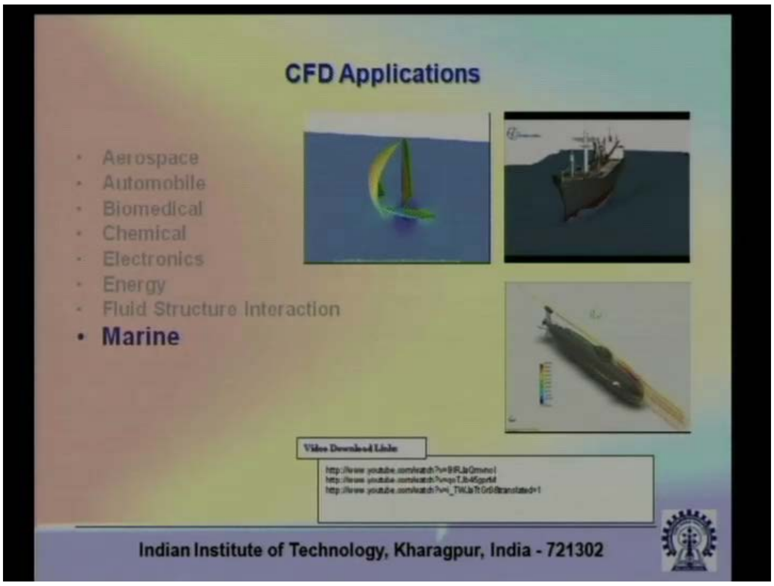
Now, when you have fluids, many times we forget that fluids interact with their boundaries and that give rise to fluid structure interactions. So, when we have fluid structure interactions, it is important to model the CFD with a structural model. So, CFD is basically a fluid dynamics model, but you should also have a model for solid mechanics, a structural mechanics, and there must be an interaction between these models, which can give rise to interesting fluid structure interactions.

Look at this example. So, here there is a balloon which sort of interacts with the surface and tends to get deformed as it interacts with the surface. So, there is a model of the surface because it has its own elasticity and stiffness. And the fluid filled balloon has its own modeling, the balloon has a membrane, so it has also its own flexibility. So all-in-all, apparently it might appear to be very simple problem, but it is actually a very complex problem in fluid mechanics.

In this example what is shown is that there are dolphins which are sort of interacting with a pool of fluid. So, dolphins are sort of models. See, CFD is all about modeling, it is not that somebody is representing the real dolphin, but representing it may be like a flexible membrane. So, you have a flexible membrane that represents or that models the dolphin artificially. And that sort of interacts with the fluid and creates a deformation in the fluid itself as well as there is a deformation in the body of the dolphins. So, that their interactions it is a very interesting problem and that this type of interesting problems can also be addressed through CFD.

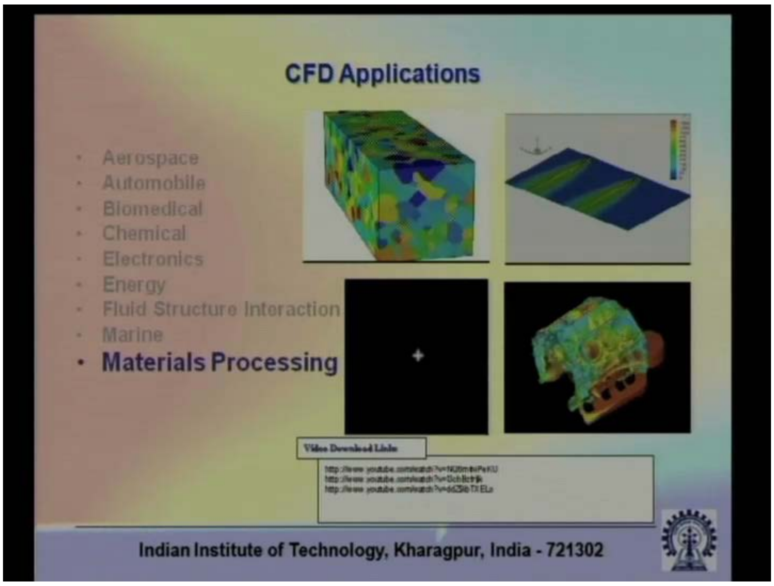
In general, fluid structure interaction problems talk about some structures which sort of have some vibration or motion, and it interacts with an insufficient fluid which is there in the surroundings, and how they interact eventually with each other. So, the fluid interacts with the structure, the structure interacts with the fluid. It is not a one-way coupling, but it is a two-way coupling, that is very important. Otherwise, one can have a separate model for fluid and a separate model for solid, but whatever is the disturbance that the fluid gives to the structure, the structure has a deformation on the basis of that. That deformation is fed back to the fluid and that modifies the velocity field within the fluid. So, it is a two-way coupling and a dynamic coupling. And the great challenge is to actually take into account all the intricacies of these dynamics, while modeling fluid structure interactions.

(Refer Slide Time: 12:01)



CFD has lots of applications in larger scales also, like marine applications. So, you can have a flow past ships and boats and even submarines, and though these are of much larger length scales than many of the examples that we have seen till now, but like these are still very important, and huge component of the marine design now these days relies on the use of CFD.

(Refer Slide Time: 12:33)



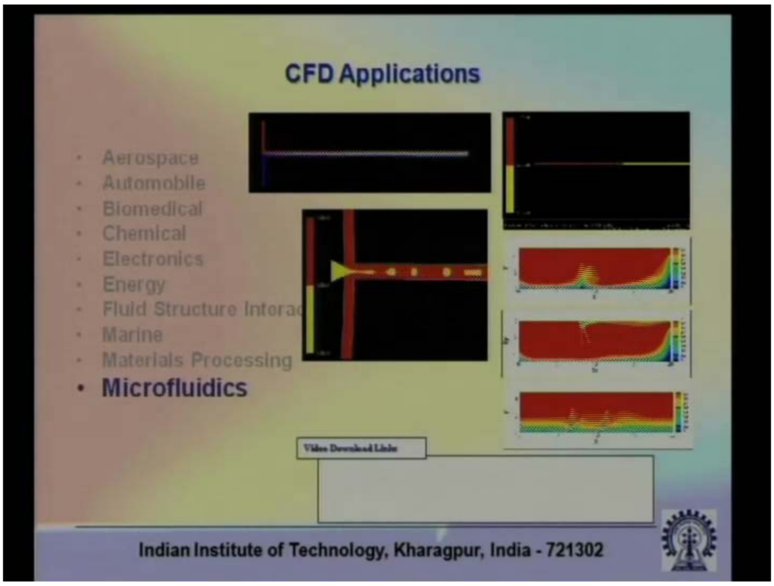
CFD has lots of applications in materials processing or manufacturing. So, like this is an example where one is trying to simulate the grain growth in a material, which sort of the nature of which gives rise to the material properties. So, this is an outcome of a CFD simulation of how the grain grows. Of course, this always need not be a deterministic simulation, the one can also use stochastic simulations like Monte Carlo simulations and so on. So, CFD need not always be a deterministic solution, it need not always give a deterministic set of approach, but there are stochastic approach is also in CFD, like in this example one such approach is used. This is an example, where one is interested about the transport phenomena during a bath-welding process. So like, you are interested to create a joint by bringing two materials from the two sides and that bath-welded joint is sort of like, what is the transport phenomena there, that is sort of represented in this view graph.

Here, what was shown in the picture was the growth of a dendritic crystal. It is a solidification of a dendritic crystal from a melt, and you can see interesting patterns as the material is being processed. And you can also have interesting applications in mould filling. So, if you are filling a mould with a molten metal and then you can see that, like it is very important to know (( )) that how well the mould will be filled up because of a particular filling process. Because if the filling process is not appropriate, it can give rise to casting defects, and those such kind of defects are not wanted, and one cannot do lots and lots of trials and experiments because that will be hugely expensive. So, if CFD gives a sort of a guideline, that these are the processing parameters that are required to have a very efficient mould filling process, then that sort of can optimize the process and make sure that defects are minimized and so on.

So, in processing of materials one faces many challenges. For example, like in many cases, we require that the product should be as homogeneous as possible. Now, what happens, materials have impurities and when there is a fluid flow these impurities are getting transported from one point to another with the flow, and many of the materials actually have to undergo these process, because many of the processing of materials involve the transport of molten material.

So, if you have a solidification process, that means prior to the solidification the material was there in a molten state, just like a casting process. Now, if you have impurities, the impurities will be transported by convection from one point to another point with the molten material. Now, once it solidifies, whatever was the impurity at that particular location it will be frozen at that location and it will give rise to a distribution of species across the material. And if this distribution of species is undesirable, then that will give rise to undesirable mechanical properties. So, to know that how to impose a control over the distribution of species that are being added to a particular system, one has to sort of assess the velocity and the temperature field and the species concentration field within the domain, which constitutes the domain on the materials processing application.

(Refer Slide Time: 16:21)



Now, if you consider applications in small scales, there are interesting applications of CFD in micro-fluidics or micro-scale fluid flows, I mean these are some interesting examples. So, here what is the purpose? The purpose is to mix two different fluids. Now, micro-fluidics deals with studies on fluid flows at small scales in channels of micron or sub-micron dimensions. In such small length scales, usually it is not possible to take the advantage of turbulence, because Reynolds number is very small. So, if you want to mix two streams, there are different ways in which you can do that. One is by clever designing of the channel geometry, that is not shown in this figure. But what is shown in this figure is that you can also try to have a good mixing by using a pulsating flow. And with a pulsetized flow input the objective is to create a good mixing between different fluid layers. In micro-fluidics, one also has important studies considering or concerning droplet dynamics.

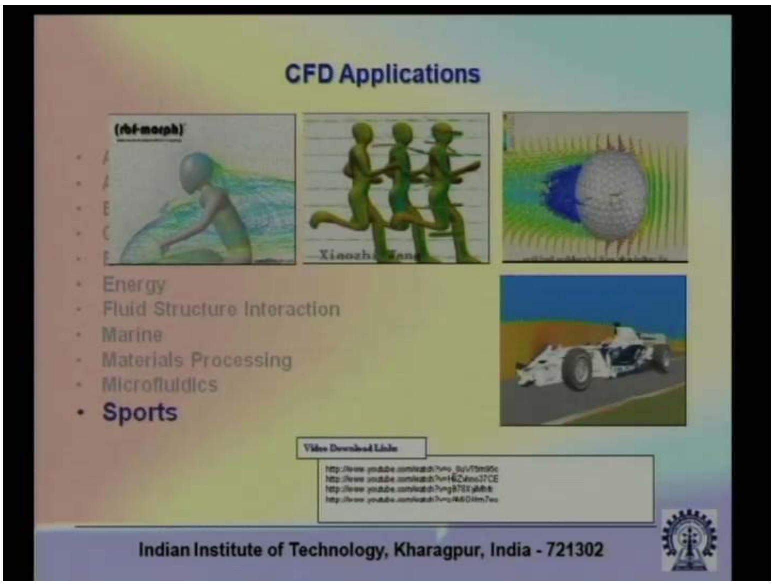
So, this is an example, where in a junction, this is a t-type of junction because of sheer droplets are produced. And then droplets are elongated and sort of compressed and so on.

So, they are deformed and it is very important to see that how these droplets evolve. Because many times you put reactants with these droplets and allow reactions to get performed in small scales, those are called as micro-deactors. And therefore, it is very important to understand that how the droplet dynamics evolves in small scales.

Now, in micro-fluidics also fluid structure interactions are important. This is an example, where you have an oscillating flap located within a micro channel or a small channel. So, what this flap tries to do is, it tries to oscillate with a particular frequency and amplitude and there is some material which wants to undergo a reaction at the wall. So, this flap what it is trying to do, it is trying to force the material on the wall at certain locations where the reactants are kept. And it is not only at one single spot where the reactants are kept, there are different locations where these are kept. So, one needs to optimize the movement of these flaps, so that all the reactant locations get the sample for reaction. And these flaps essentially act like a combination of a mixer and a pump. And because mixing is not so easy in small scales, this is one of the strategies by which one can have a better mixing by using mechanical flaps in small scales.

Of course, there are many other applications, but our objective is not to go deep into any application, but to just appreciate that different fields have different challenging applications of CFD, the whole objective is not to go deep into any particular field at this particular stage.

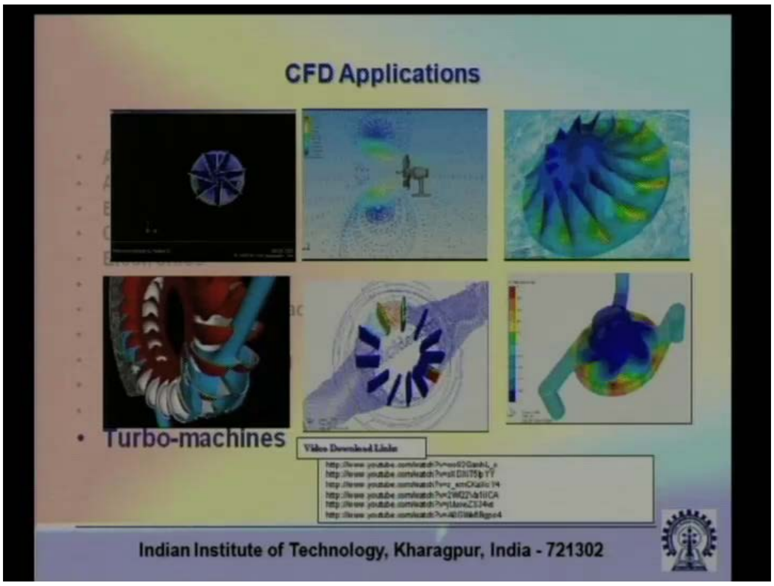
(Refer Slide Time: 19:34)



Now, just if you come to some fields of entertainment like or sports may be, there is a huge application of CFD in sports. So, for example, if there is somebody who is sort of driving a racing car, then how the fluid flows past that. It is very important because again in such applications it is a sort of a very important idea how to minimize the drag, and then to do that one needs to design the car. And to design the car one needs to taste it. And of course, if one fabricates it in a real size all the time and does the testing, then it may get over expensive. So, before the actual fabrication, during the design stage CFD is very commonly used. So, even when human beings are running, so how the fluid flow takes place around human beings when they are running around the body contour. Because that sort of dictates that what is the total drag force and the running posture interacts very nicely with the fluid flow, in a way that one can have a technique by which one can minimize the drag force under certain conditions.

The dynamics of sports balls is also very important, like golf ball, cricket ball all these things, that how fluid flow takes place around these. Because it is the extent of turbulence or the nature of the flow, whether it is laminar or turbulent, that sort of many times dictates the movement of the ball. So, you can have the swing of the ball and you can have other sorts of, other types of lateral movement.

(Refer Slide Time: 21:26)

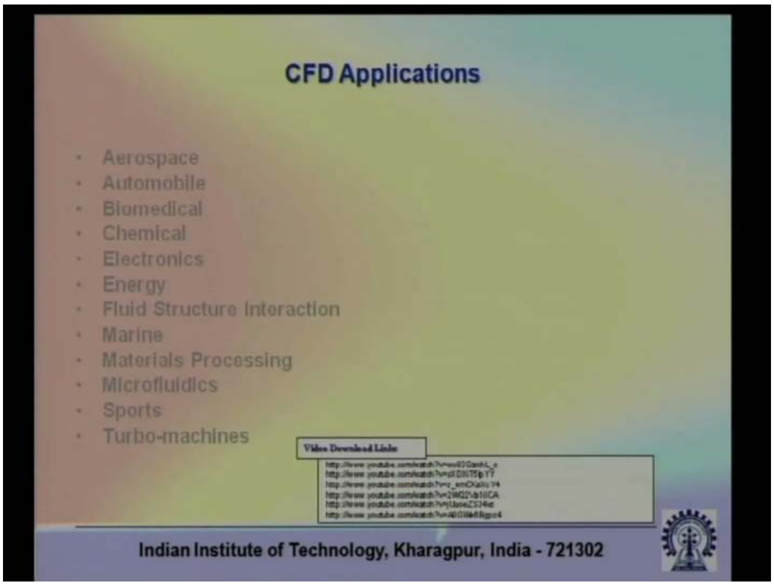


So, in sports CFD is very extensively used for scientific purposes. In other applications of mechanical engineering, like the one of the traditional applications is turbo machines.

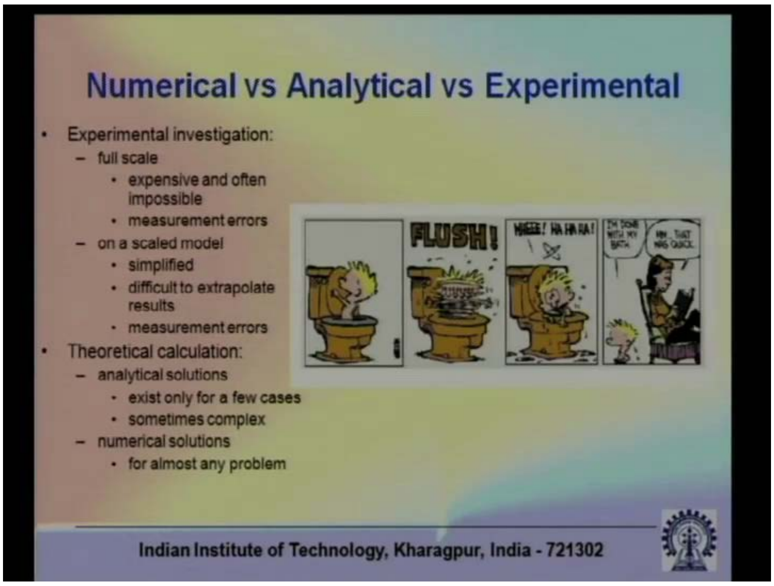
So, in turbo machines CFD is very commonly used for design of like blades or flow passages, so to say in general, those may be blades or impellers. So, you can see that these are wheels of Helton turbine or the buckets. So, you can see that you have a wheel and then on the top of that, I mean on the periphery of that, you have buckets and fluid jet is interacting with the bucket and this entire process is being simulated by CFD.

So, you can have CFD applications in turbo machines, where you can sort of access in a great details of how fluid flow takes place, for example, across the impeller of a pump. Now, this is very important because like analytically it is virtually impossible to analyze such flows, because the blade passages are very complicated. It is not as simple as like flow over a flat plate or flow in a parallel plate channel or flow in a circular pipe, because of such complicated geometry across which the fluid flow is taking place, it is almost inevitable that one has to go through computational roots for solving such problems.

(Refer Slide Time: 22:50)



(Refer Slide Time: 23:11)



So, we have seen some applications of CFD and we have now developed an idea that in these applications why CFD is important, but the bigger issue or the bigger question is, is CFD inevitable; that means, is it so that we require CFD in all cases. That is one interesting question that we would like to answer. Because many times, because this is a CFD course, we will be having a bias towards CFD. So, we will say that yes CFD is necessary. But to be honest, let us try to answer this question that, is CFD necessary in all cases. So, that is the second question that we wanted to answer; that is, when CFD and why CFD. So, we have understood by this time that CFD is about numerical simulation of some governing equations for transport phenomena; that is, in a summarized form. But to solve the transport phenomena problem, you could also have an analytical approach and you could also have an experimental approach. So, these are alternative approaches to CFD.

So, question is that when should we prefer other approaches or when should we prefer CFD. Let us try to look into that. So, first of all experimental investigation, we should keep in mind that there is no substitute for experimentation, because seeing is believing. So, there are many idealizations that we can sort of do, while we are doing a numerical modeling or analytical modeling, but when we do experiment and see something is happening, that is the reality what is happening.

But having said that, there are certain important limitations or restrictions that can go with experimental investigation or experimental modeling. What are those? So, when we say modeling it can be both experimental modeling and numerical modeling. We are now focusing on experimental modeling. Experimental modeling in full scale is expensive and often impossible. So, like, for example, if you are interested to study fluid flow in a steel making vessel. So, in a steel plant, in a steel making vessel, there is a very interesting flow pattern, because you may have jets of oxygen being input to the fluid. You can have the jets coming from both the top and from the bottom. And so there can be multi-faced flows in the system. So, it can be a very complicated flow and eventually the final property of the steel is a strong function of how the flow is taking place in the steel making vessel. But imagine that you have a molten steel at such a high temperature plus the medium is also not transparent. So, you cannot have a full scale model with a real fluid under those applications.

So, it could be possible in this case, it is in the real case it is not possible to study the fluid flow exactly in the system. There are certain examples where it is possible, but it is very expensive. So in general, full scale modeling may be expensive, often impossible. Expensive full scale modeling means you are designing an aircraft. So, before designing or before coming up with a final dimensions, if you want to make a lot of trial by experimenting with a real sized aircraft, then that is very expensive, I mean, one should not go for such a design because it will involve a lot of expense. Plus just like any sort of any tool has its own errors, experiment also has important errors, which are related to measurements. So, it is not that seeing is, believing, yes seeing is believing, but measuring is not always believing. So, whatever you are measuring may have lot of error associated with that.

Now, many times because of limitations with the full-scale modeling, one goes for a scaled model. The scaled model may be smaller than the actual model, it may also be larger than the actual model, depending on whatever is convenient. Now, scaled model should satisfy certain important objectives. That once you have a scaled model, you should not scale it up or scale it down in such a way that the flow physics has changed. For example, say you are interested to study flow in a micro capillary. Now, you say that well it is difficult for me to do experiments in a micro capillary, so I will do same experiment in a pipe, which is sort of geometrically similar version of the capillary, but hugely expanded in dimensions.

So, geometrically you can maintain the similarity, but the physics of the problem might change. Because in the micro capillary, surface tension force is very important, but when you go to a large pipe, surface tension force may not be that important. So, when you are considering a scaled model, one has to keep in mind that at least the physics does not change. But many times one has to make a compromise, and once that compromise is made it becomes a simplified version of the reality, but not the reality itself.

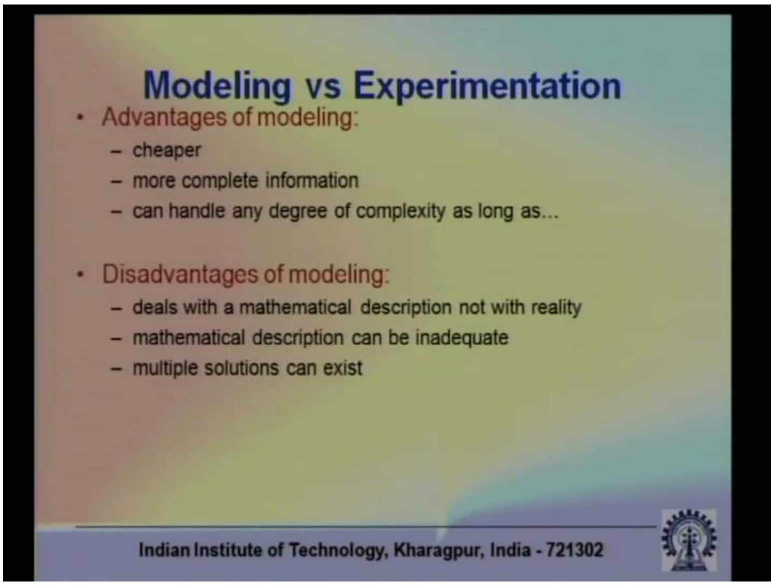
Sometimes it is very difficult to extrapolate results from the model to what will happen for a prototype, because of incomplete similarities. Because many times, exact all sort of similarities; that is, the physical similarity, geometrical similarity, kinematic similarity, dynamic similarity, all these similarities may not be properly maintained, and then it may be difficult to extrapolate the results. Plus measurement errors are there even in the scaled model, not that in scaled models measurement errors are not present. Next, so this is about experimental investigation. So, experimental investigation has many advantages, here we have only itemized the critical concerns, it is not that we have considered just the positive points, but we have considered it very critically, where the constraints are also taken into account.

Theoretical calculations, if you come to theoretical calculations, theoretical calculations for the same problem will have two sorts of approaches. One is an analytical solution. Analytical solution, if it exists there is nothing like it, because it is exact and it gives exact point to point variation without any approximation. But the question is, does the analytical solution exist for the case that you are interested for? Well, for most of the practical cases analytical solution does not exist. So, it exists only for a few cases, those few cases are very important. See, we will see that CFD cannot stand alone by itself, it requires a good help from experimental as well as analytical solutions, because you require to benchmark your solutions. Say, you are solving a very complex problem of fluid flow and you have written a code or you have developed a code, how do you validate that? You can validate that against some problems, which are either standardized in their solution by analytical means or there are benchmark experiments with which you can compare. So, you cannot really discuss CFD devoid of the considerations of analytical solutions and experimental solutions, that is why we are bringing these into perspective. So, analytical solutions exist only for a few cases. And in cases where they exist, sometimes they become very complex.

So, I have seen students by looking into analytical solutions of that problem so easily, sometimes they get driven away from the problem that they never come back for solving the problem. And the reason is that sometimes the expressions are so big and so cumbersome, as if like wherever inside those expressions some physics is hidden, it is very difficult for beginner to appreciate that in many cases. But if analytical solution exist, there lies its own elegance and its own beauty and it needs to be appreciated and it can be used as a benchmark for more complicated numerical solutions.

Numerical solutions exist for almost every problem. Of course, whenever you gain something, you lose something also. When you say that numerical solution exists for almost every problem, what we mean by numerical solution is that you are getting solutions at discrete points in the domain, not continuously at all locations. So, the continuous nature of the solution gets sacrificed, but at the expense of that you at least get solution for even complex problems.

(Refer Slide Time: 31:56)



So, if you compare modeling versus experimentation, what are the advantages of modeling? Advantages of modeling are as follows, they are cheaper than experimentation in many cases. For example, if you have developed a particular code or you have got a code which can be used for a particular problem, then you can simulate it for different cases and for that expenses do not mount up. So, it is the same code that you are running with different parameters. Of course, you are incurring computational time. And if it is a highly sensitive code to computational effort; that is, you are for example, doing a direct numerical simulation for a turbulent flow problem or solving a problem in a highly paralyzed computing environment, then it involves a lot of cost, it does not mean that computation is free. But only thing is that many times it may still be less expensive than the corresponding trials, which one could have done with experimental investigation.

With modeling you get more complete information. Why? Say, with experiment, you do not really have a scope of getting all the details of all the variables that you were looking for. Because for example, you are interested for some flow measurement, temperature measurement, velocity measurement, of course, you can do that within the complete flow field by numerical simulation, you can also do it also by experimental simulation. But if you were interested to do that, it will involve a huge expense. Not only that, in many cases it is ruled out as we said. For example, if you interested in a flow visualization for a molted steel, say, if you want to get velocity profiles within that or velocity vectors within that, numerically you can generate, experimentally it is not possible. One can handle any degree of complexity, that is it can be a very complex problem and one can handle it, but there are important buts. What are those buts? Like, when you are handling a very complex problem, many times what is not known is a correct description of the boundary condition. So, you see a numerical solution is as good as the input that is going with the solution. So, if you are solving a problem numerically and you are getting a solution, whether you need to believe the solution or not, it is up to the level of the input. So, what we mean by the level of the input? The input mainly deals with the input of thermo physical properties in CFD applications, like for example, input of properties as a function of temperature.

So, if you are giving the properties as a function of temperature and composition very accurately, then that is a very good thing, but where from you will get that. So, you must have reliable sources of property data, that is one thing. The other thing is boundary conditions. So, many times we give boundary conditions in numerical simulation which are not correct representatives of physical reality, like for example, we say isothermal boundary conditions. So, in reality, whoever has done experiment will understand that it is one of the toughest things to maintain an isothermal boundary, even in a very simple case.

So, whatever we are sort of planning to be isothermal, in the real experimental case, it may not be maintained as isothermal. So, that is one example. So, the understanding is that there can be lots and lots of uncertainties with regard to the boundary conditions. And one has to consider these uncertainties very seriously, because sometimes the problem may be so non-linear that with a slight change in the initial condition, with a slight change in boundary condition, the solution may change, the solution may bifurcate to an entirely different one from what you are interested to get.

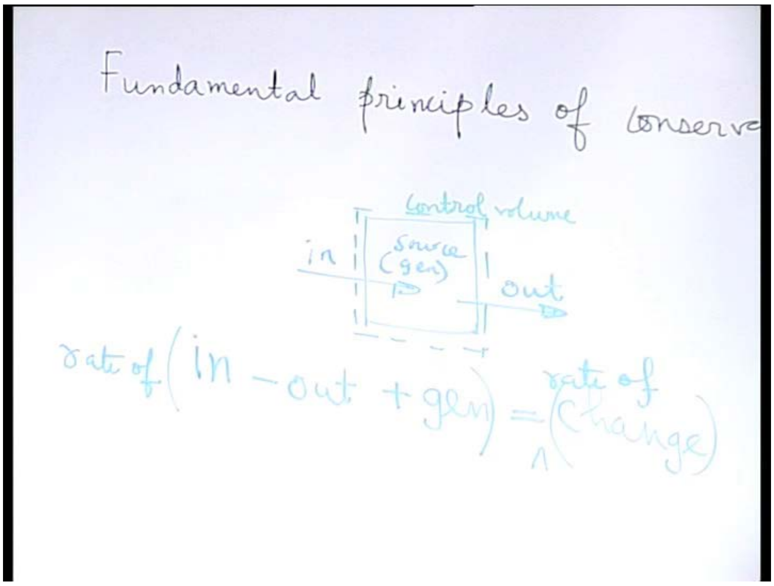
And that is where it can be very critical. So, we should not basically be so emphatic and say it is disadvantage of modeling, but may be one can say some limitations that modeling deals with the mathematical description and not with the reality. So, when you say a boundary condition, it is a mathematical description, that mathematical description may correspond to the reality, may not consider also to the reality. So, one has to keep in mind that it is your responsibility as an analyzer to make sure that it deals with a reality as much as possible to whatever extent possible.

But mathematical description can be inadequate. So, how it can be inadequate? So, we have considered the property data, we have considered the boundary condition, what about the governing equations themselves? So, are the governing equations representing the correct physics that you are looking for? May be or may not be. May be the correct physics that you are, or the physics that you are trying to sort of capture, cannot be captured by the equations that you are employing. So, then no matter whether you are using correct properties or correct boundary conditions, your solutions will not be the desired ones, simply because you are not representing the correct physics. Sometimes, modeling particularly for non-linear problems can be tricky, because non-linear problems may have multiple solutions. And by modeling, you might get each of these solutions by having different initial guesses. For example, if you are using iterative schemes for solutions, so the question is what would be your correct solution. Because may be, when you are doing experiments, you will get only one solution during one experiment. So, how do you relate your experiments with the modeling, that is a very important concern so to say.

So, to summarize this discussion on modeling versus experimentation, we can say that there is actually no substitute for experimentation, but there are limitations associated with experimentation, maybe because of expenses or sometimes it is because of the inability to capture many things during experiments, like for example, capture the velocity field in a non-transparent material flow, as an example. So, because of these limitations one cannot do experiments on a trial basis. So, if one is interested to go for design, then the best route for solving a problem may be like this, that first make a model of the problem, and then try to approach the problem mathematically, come up with some solutions, either numerically or analytically depending on whatever is more convenient. Of course, if analytical solution exists that will remain to be more convenient, but if it does not exist, then one has to go for numerical solutions. And from those solutions, reduce your number of trials, so that you come up with only a few sets of data with which you do final set of limited and very controlled experiments.

So, study of CFD is not to make us go away from experiments, but actually to help us in doing better experiments, so that we can get a domain where we have very restricted and very highly refined sets of data towards a good experiment design. And then on the basis of that design, if you do experiments, that will give us a good feel of the reality what is happening. So, then with that experiment we can compare some of the CFD output, some of the CFD output we cannot compare with experiment, because such details you may not get from experiment. But if other details are verified, then of course we can say that the CFD model is representing the reality in a very nice way. And then you can do many sets of parametric studies and come up with nice conclusions which sort of may not be possible with just experimentation. So, many times in old days, experimentation was like a hit and miss type of trial. So like, it is just like an alchemist, where one is mixing A with B to see that whether C is becoming gold. Those approaches towards experiments were very ancient. Now-a-days approaches to experimentation is very sophisticated. And in fact, CFD plays a big role towards good experimentation. Now, next what we will do is. So, we have seen that, what is CFD, and sort of why we are doing CFD, and how do you compare modeling with experimentation, and when to do experimentation, and when to do modeling and so on. Now, next what we will try to do is, we will try to see that what are the basic principles that govern the implementation of CFD. And when we say that what are the basic principles that govern the implementation of CFD, we have to keep in mind certain things.

(Refer Slide Time: 41:36)



So, what are those certain things. So, those certain things are fundamental principles of conservation. So, fundamental principles of conservation are the principles, which govern the basic equations that we commonly use for CFD. And not only CFD, but even analytical fluid dynamics. So, to understand that what are the basic principles that go behind, we will try to learn these principles of conservation in a very generic manner to begin with, and then we will try to apply it for the transport of mass, momentum, heat like that.

To do that, we will first see that what essentially does, conservation talk about. We will start with an example, that example is not falling in the purview of heat transfer, mass transfer or momentum transfer, but just to make sure that you are not very disinterested, we will consider money transfer. So, let us say that you have a bank account and you are interested to transfer some money to the bank account and withdraw some money from the bank account. So, let us consider ancient times, not the present internet banking system. Let us consider that it is a traditional era, when you are going to the bank and you are opening a bank account. So, you are depositing some money to the bank account and your bank account is activated. Now, say, every month you get some salary, scholarship, whatever, and there is a direct transfer of money to your bank account. So, your bank account is getting enriched. Now, suddenly one fine morning you feel that you need to withdraw a good amount of money.

Remember we are talking about old age, when it is not an ATM by which you are operating. So, you are physically going to the bank and you are withdrawing some money. After withdrawing the money, you are checking that what is there in the bank account. In fact, this same procedure you can do with the ATM also, it is not that you cannot do, but just for simplicity we say that you are checking the bank account in which some money has gone into it, to begin with, you have withdrawn some money. So, you have some money in, you have some money out, which you have taken out. And you are interested to see now that what is the balance that you are having now. So, you can see that whatever was the money in minus whatever was the money out is not the balance that you are getting, you are having the more balance than that, because the bank is generous to give you some interest. So, there is some source of your extra funding and that is by interest provided by the bank. So, this source we may also call as generation.

So, we can say that, in minus out plus the generation equal to the net change of money in your bank account. So, that is the statement of conservation of your bank account management or bank balance. Believe it or not, this is the same principle which governs the behavior of conservation of any other physical variable, maybe mass, momentum, energy or whatever. So, for many of those, what we do is, we express this as rate, rate of in minus out plus generation is equal to rate of change.

And there are principles which sort of try to execute this conservation principle, I mean, there are mathematical formalisms which try to execute this conservation principle and apply it to the conservation of mass, momentum, energy and so on. So, this bank account which we have just considered as a specific example, if you consider it as a more generic case, you can consider it to be a region in space across which different quantities can flow, the quantity here is money, but it can also be mass, momentum, energy like that.

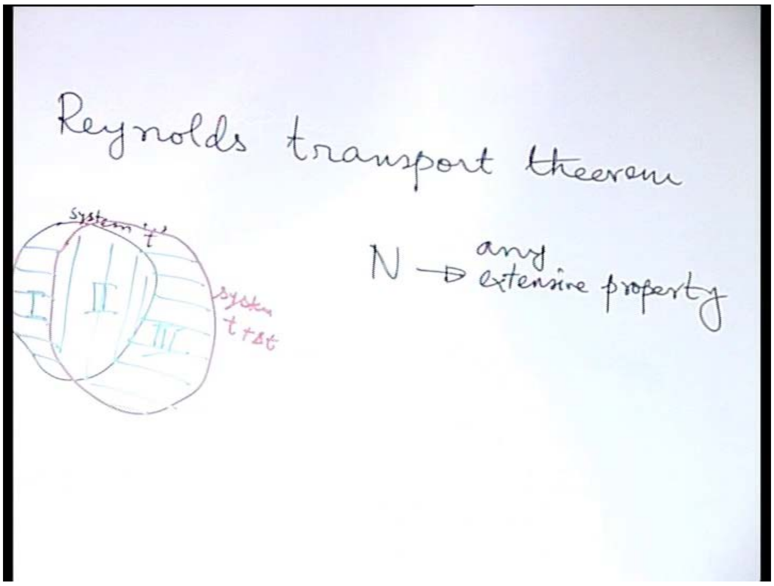
So, this by definition is a control volume. So, across this control volume there is transport of some quantity. So, this control volume what is that? It is a identified region in space across which matter and energy can flow. So, when you have a control volume, we can write a balance for the control volume, true. Now when we write a balance for the control volume, where lies the difficulty? The difficulty lies in using some of the basic equations in mechanics, which are not inherently developed for control volume.

For example, Newton’s laws of motion, those are not inherently developed for control volumes, those are inherently developed for identified sets of particles. So, those equations are inherently suitable for a Lagranian description; that is, you have identified motions of particles and for each particle you are writing equations of motions. So, you have a trajectory of each particle which is governed by the Newton’s laws of motion. On the other hand, this approach is called as Eulerian approach, where you have a Eulerian control volume and you are interested to study the transport behavior across that.

All of us know that for fluid mechanics, it is more convenient to use the control volume or the Eulerian approach. Because fluid is continuously reforming, if you are identifying some particle in a fluid, then what will happen? There will be numerous particles which will be evolving in very complicated ways, until and unless the flow is very simple. On the other hand, if you just focus your attention, as if you were sitting with a camera and focusing attention on a specified region, you are not having to track the particle, but what you are just seeing is, what is happening across your focused region.

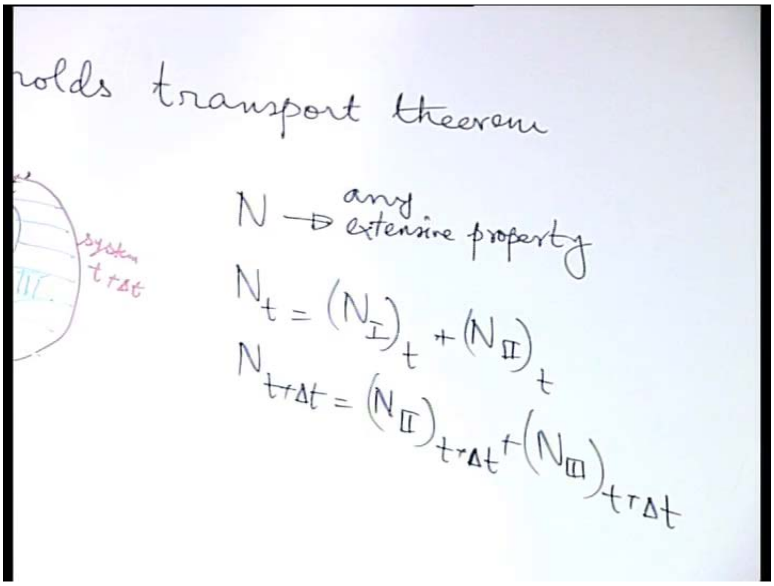
So, the Eulerian approach is more convenient, only problem is basic equations of mechanics and thermodynamics are not originally developed for Eulerian description. So, we must have a transformation from Lagranian to Eulerian description, so that we can use that Eulerian description, at the same time we can use the basic laws of mechanics that were originally developed in the Lagranian frame. And that description is given by the Reynolds transport theorem, which we will just revisit.

(Refer Slide Time: 49:57)



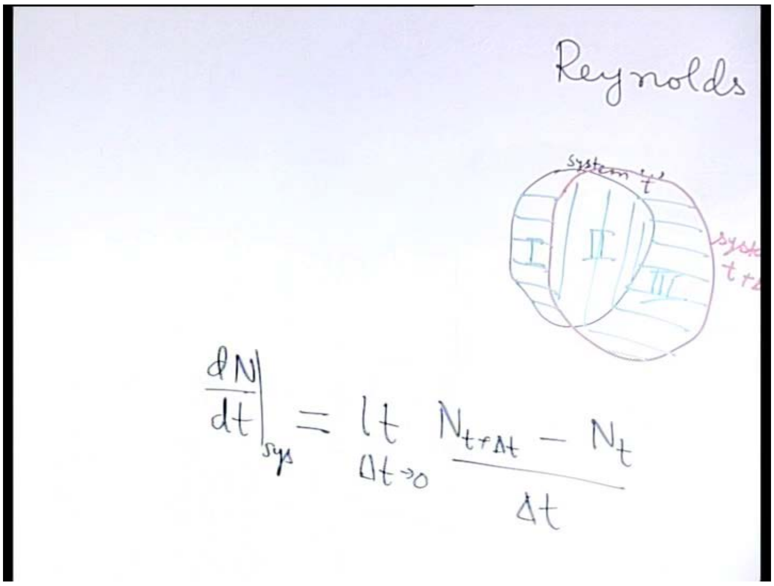
Let us say that we have a system. System is essentially an identified collection of particles of fixed mass and identity. So, it is a representative of the Lagranian description. So, we have a system at time t, we consider the same system in a different configuration at time t plus delta t, where delta t is small. Actually because delta t is small, these two configurations are almost coincident, but just for clarity we have shown them distinctly.

(Refer Slide Time: 52:42)



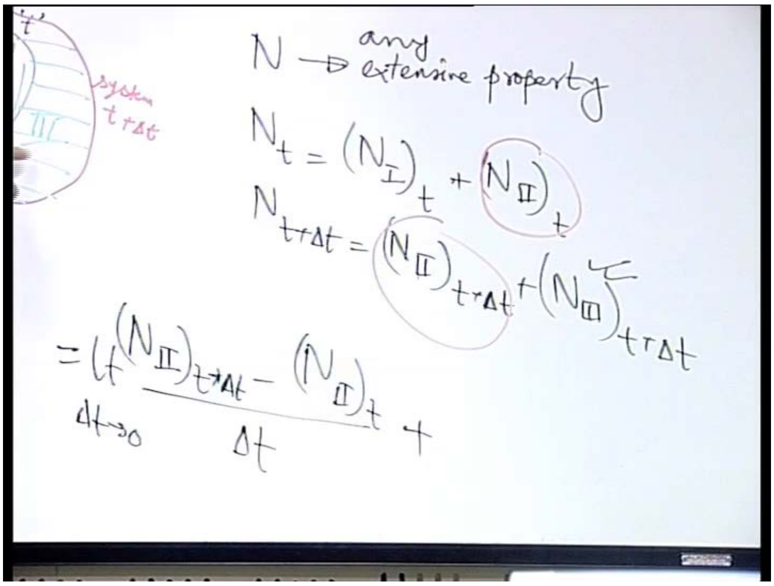
Now, this overlapping of the systems at two different instances gives rise to three different regions. One is this region 1, then this region common region 2 and the third one is the region 3. Let us say that we have some property N, which is extensive property, any extensive property. So, N at time t is N occupying the region 1 at time t plus n occupying the region 2 at time t, because at time t it is region 1 plus region 2.

(Refer Slide Time: 52:18)



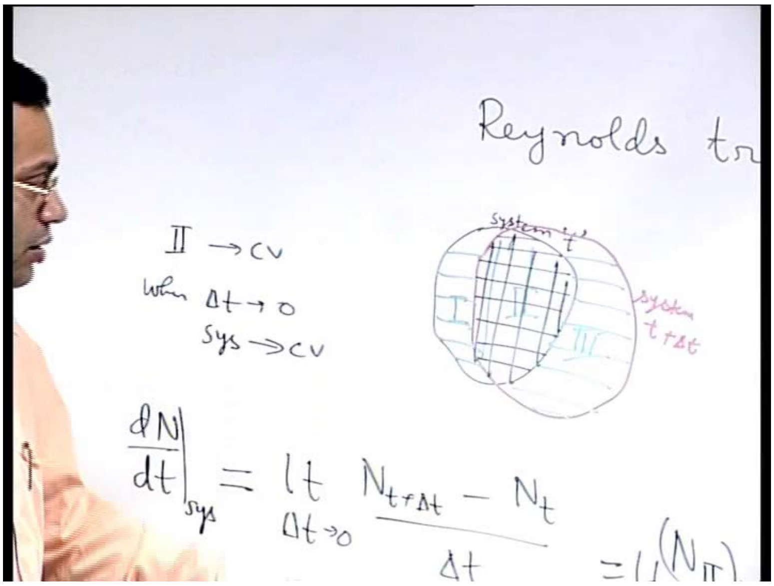
N at time t plus delta t is N at 2 at time t plus delta t plus N at 3 at time t plus delta t. We are interested to find out what is dN dt of the system; that is, the rate of change of N for the system. So to do that, we can write dN dt of the system is equal to limit as delta t tends to 0 N at t plus delta t for the system minus N at t divided by delta t.

(Refer Slide Time: 52:53)



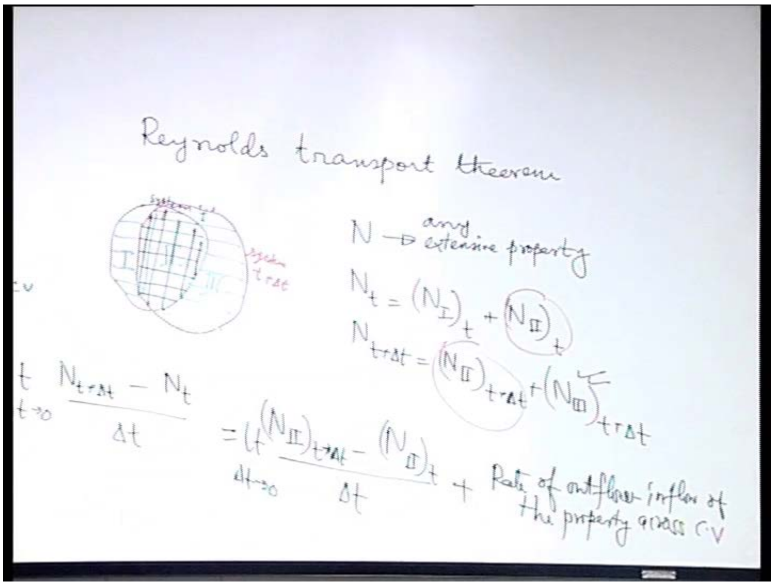
So, N at t plus delta t, you can see that it has combination from region 2 and region 3, out of these region 2 is common. So, we can leave that apart and write it or break it up into two different terms. One is N at 2 at t plus delta t minus N at 2 at t by delta t in the limit as delta t tends to 0 plus, we can write something bit qualitatively, plus we have this term N at 3 at time t plus delta t divided by delta t, what is that. See, at the region 3, whatever is the property, that property is ready to leave the control volume, what is the control volume here?

(Refer Slide Time: 53:55)



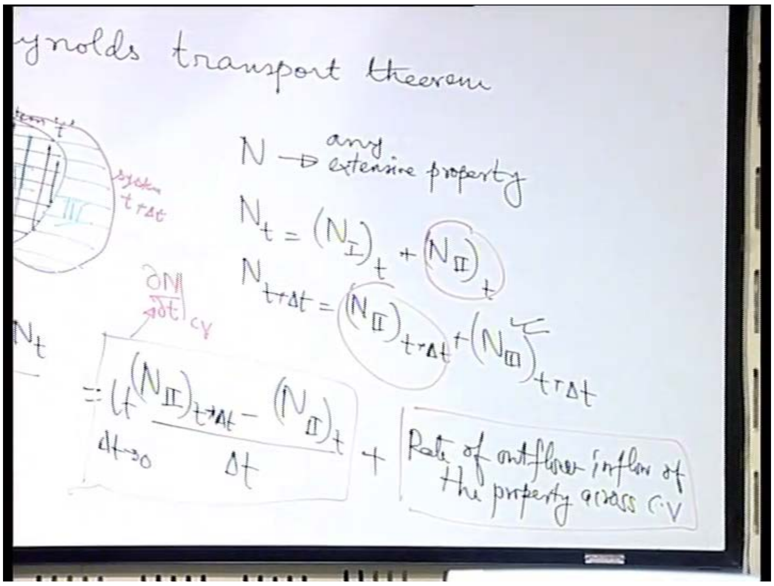
Remember when delta t tends to 0, what is your identified region in space? When delta t tends to 0, this 2 is your identified region in space. So, 2 is your control volume and when delta t tends to 0, your system and control volume almost coincide. Because these are almost merging configurations, but your control volume is 2. So, when you have 3; that means what, it is ready to leave the control volume and when something is there at 1; that means, it is something ready to enter the control volume.

(Refer Slide Time: 54:35)



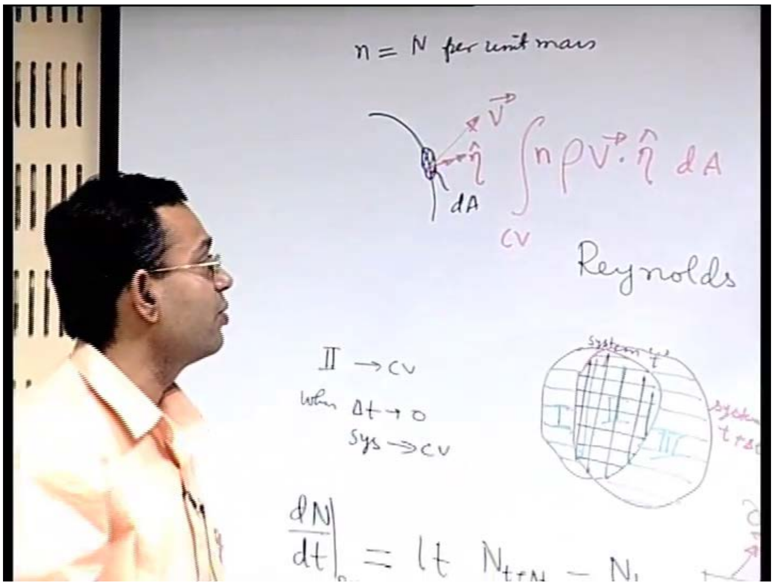
So, we can say that this is nothing but, plus rate of outflow minus inflow of the property across the control volume. So, we can write this, rate of outflow minus inflow of the property across the control volume in a bit of a mathematical way.

(Refer Slide Time: 55:07)



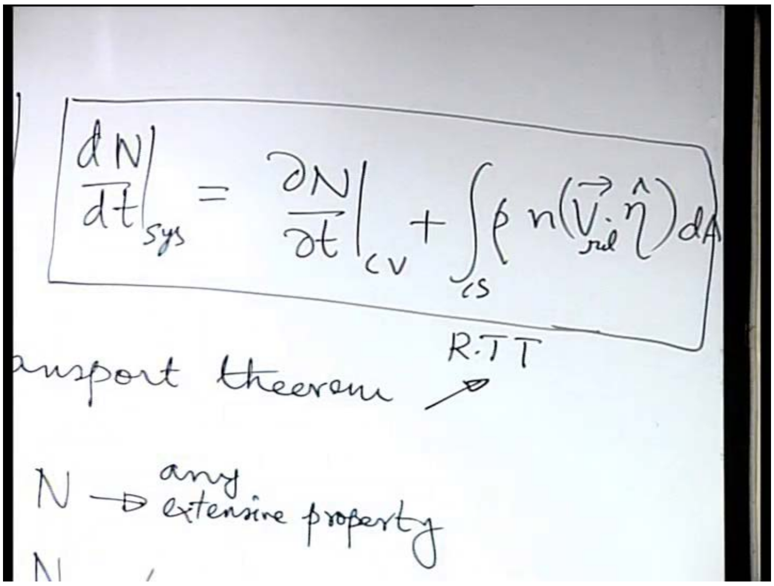
Like, just we can first write this one. This one is nothing but, partial derivative of N with respect to time in the control volume. Because here we are fixing the position and changing N with respect to time, that is why it is the partial derivative.

(Refer Slide Time: 55:40)



Now, let us look into the last term rate of outflow minus inflow. So, for that we define small n as capital N per unit mass. Now, when there is a surface on the control volume, let us say that there is a small area dA and the fluid is having a velocity of V. So, what is the volume flow rate there? What you have to do is to just construct a unit normal vector, let us say eta is a unit normal vector to the area. So, V dot eta dA is the volume flow rate. That times the density is the mass flow rate. And small n is the property per unit mass. So, n times this, is the total transport of property. So, if you integrate it over the control volume, it can give outflow minus inflow. Because if V dot eta is positive; that means, outflow if V dot eta is negative; that means, inflow. So, if you consider it over the entire control volume, it automatically takes into account outflow minus inflow.

(Refer Slide Time: 57:08)



So, with that consideration, we can finally write the conservation expression as follows, dN dt of the system is equal to… One important thing to mention is that, this V is not the absolute velocity, this is velocity of the fluid relative to the control volume, because that dictates what is the flow rate. So, we will call it V relative. This expression is a mathematical representation of a conservation principle known as Reynolds transport theorem.

We will stop here today and in the next class we will see that how to apply this theorem for various applications of conservation, and which leads to different conservation equations that we will use for CFD applications.