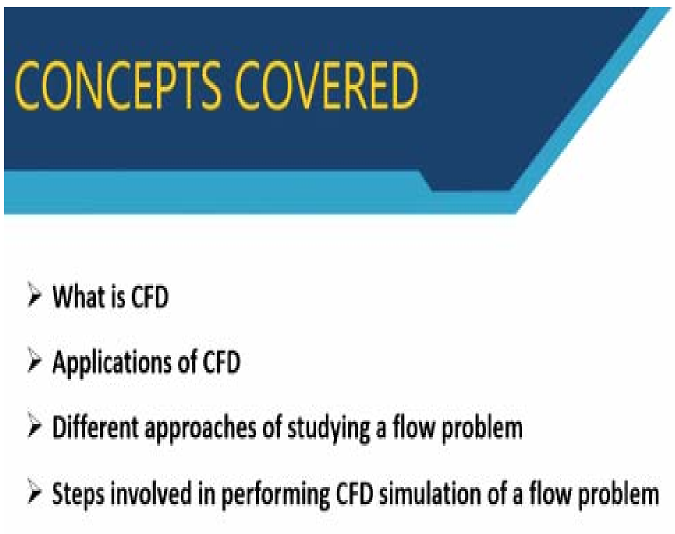
Introduction to CFD
Prof. Arnab Roy
Department of Aerospace Engineering
Indian Institute of Technology – Kharagpur

Module - 1
Lecture – 1
Brief Overview of CFD

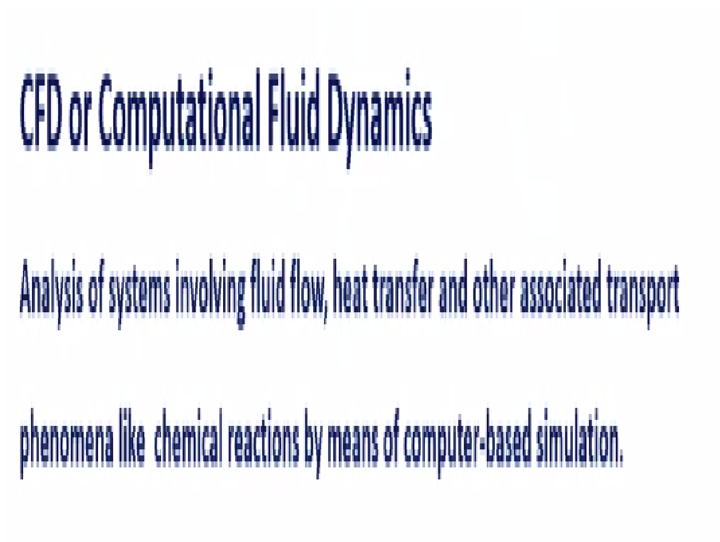
(Refer Slide Time: 00:37)



Welcome to the course on introduction to CFD. In this course, we want to study computational fluid dynamics from an introductory perspective. In this lecture, we would try to address the following issues. We would look at what is meant by computational fluid dynamics, what are the contemporary applications, different approaches that we take to study flow problems of which computational fluid dynamics is one.

When computational fluid dynamics is deployed, what are the steps involved in performing computational fluid dynamics simulations of flow problems.

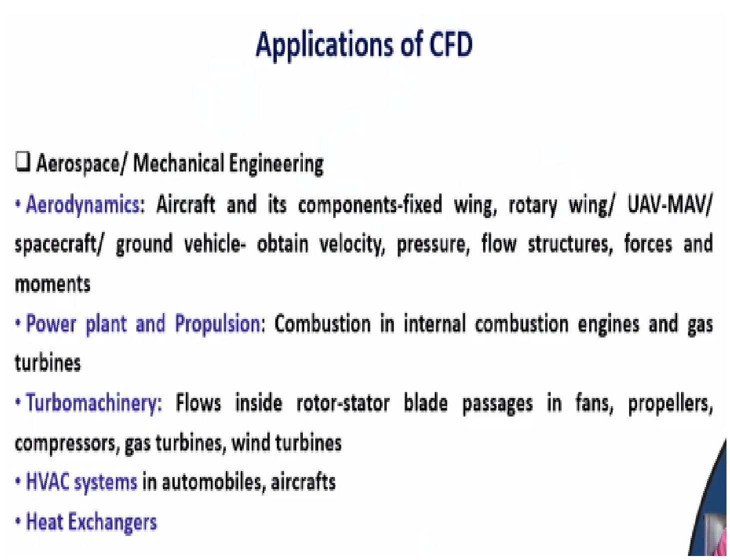
(Refer Slide Time: 01:10)



So, what is CFD? The full form is of course computational fluid dynamics, but it is not always essentially only fluid dynamics. It could involve heat transfer, it could incorporate other transport phenomena. So, this could be a possible definition. Analysis of systems involving fluid flow, heat transfer and other associated transport phenomena like chemical reactions by means of computer-based simulation.

So, when we try to study CFD, we need to understand fluid flow, heat transfer and we also need to understand how computers may be deployed to perform these simulations.

(Refer Slide Time: 01:59)

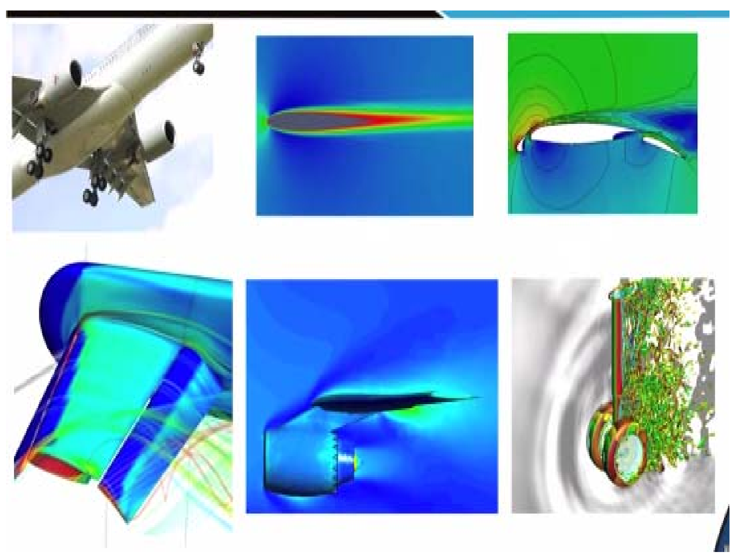


If you look at contemporary applications of CFD, there are numerous. So, in aerospace or mechanical engineering, these could be some of the applications we could look at. Aerodynamics, which is essentially the study of moving air as it flows over different bodies of interest which could be aircrafts, which could be ground vehicles like buses and cars. So, we have mentioned a few of them here.

What do we really want to study when we do a CFD simulation is we try to look at how velocity, pressure, different flow structures like vortex structures or forces and moments occur on such objects of interest. So, CFD helps us to obtain information about these different flow parameters. You could be interested in doing CFD simulations in power plants, which are used in our aircraft engines, where we have gas turbines or we could have internal combustion engines which we use in our cars.

You may be interested to look at how flows take place through turbo machinery blade passages that we find in fans, propellers, compressors, gas turbines, wind turbines. You may be interested in looking at HVAC systems and you may be interested in heat exchangers and many more applications of that kind.

(Refer Slide Time: 03:48)



Apart from aerospace and mechanical engineering, there may be applications in many other domains too, which we will come to later. Now, let us have a look at a few pictures where we see different aircraft components and how flow is occurring past them. So, on the top left you see an aircraft flying past you at an altitude and there are so many different components which are visible.

So, there is this long cylindrical body which we call as fuselage, there are wings, there are engines mounted on the wings, there are wheels which are visible near the wing, fuselage junction and things like that. So, computational fluid dynamics tries to obtain information about how flow is taking place around this whole aircraft which is a very complex geometry and also sometimes looking at only bits and pieces of the aircraft one at a time.

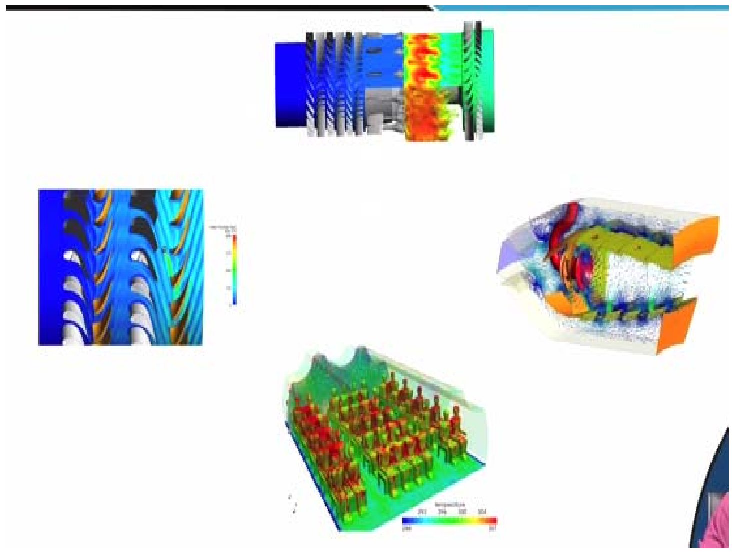
Let us say the picture that you see on your right where you find that you are actually having one section of the wing and then if you want to look at what is going on in and around the wing, you see that these colors are probably representing some information about how the velocity field is behaving around the wing. In some regions, it is high. In some regions, it is low, and the colors are expected to show that variation. So CFD helps you calculate the velocity.

It only does not calculate velocity, it could be calculating pressure, it could be calculating other flow parameter variations as well. To the right of that, you see a wing section with many components, which are apparently deployed from the main portion of the wing and then there are flow patterns visible in the gaps. So this is a little more complicated geometry which we call as a multi element airfoil.

It is a section of a wing and you are seeing many small elements in and around the main section. Likewise, you can have many other geometries around which flows can be calculated. So, this is a full wing, this is the end of the tip of the wing and these are certain control surfaces which have been deployed. Here you see an aircraft engine below the wing. Here you see a landing gear and interestingly, you can see certain features in the flow field. These appear to be like waves.

These are most probably pressure waves, which are created as the wheel juts out of the body of the aircraft when it is still flying at a significant speed and that could create pressure waves around the wheels, and these are very complicated flow structures carrying a lot of vorticity with them and these structures along with these pressure waves if they are fluctuating with time could be a source of major noise. These are of enormous importance in CFD and these days there is a lot of focus on looking at these aspects.

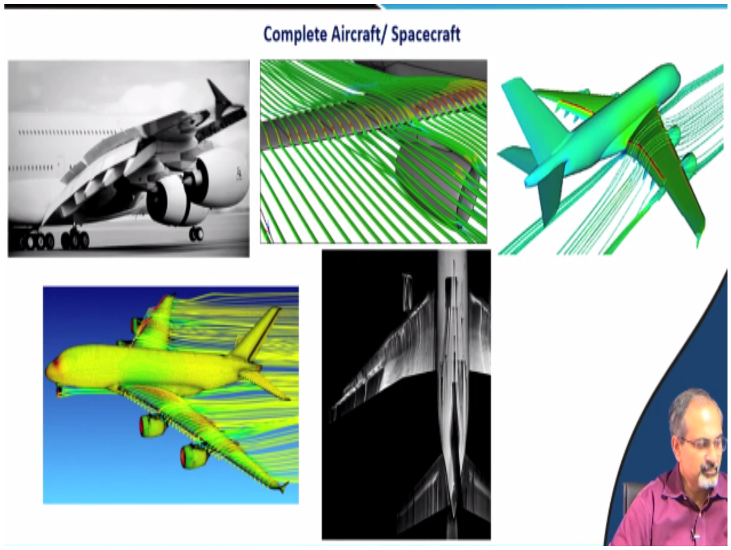
(Refer Slide Time: 07:22)



Here you see flow moving through blade passages like ones which we see in gas turbine blades, compressor blades, turbine blades and things like that. Here you have a part of a gas turbine engine where you have several stages of the compressor through which the flow gets compressed and then reaches the combustor where you put in fuel and you burn it. So, apparently the flow is on the colder side here and it becomes much hotter as it moves into this section and CFD can compute all this.

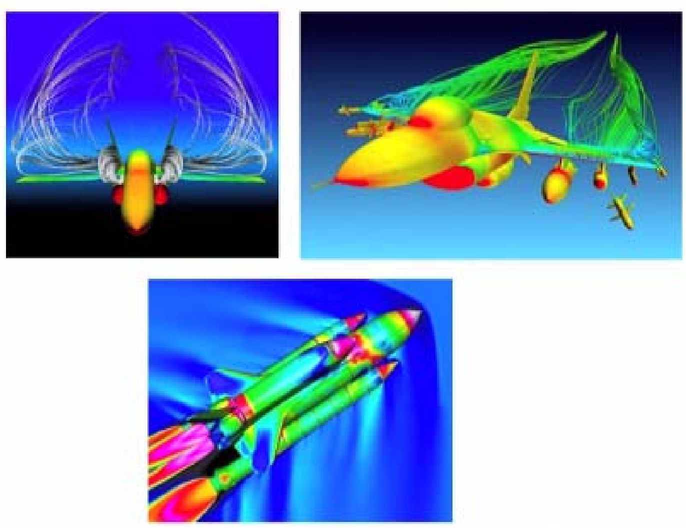
What is going on exactly inside the combustor you get a feel from this picture. So, there are very complex flow movements here and a lot of swirling flow, which helps mix the fuel and air and CFD can calculate that for you. We find passengers here seated in the cabin of the aircraft and then you can see how the air conditioning works, so that passenger comfort is ensured.

(Refer Slide Time: 08:30)



Here you see the entire aircraft with a very complicated flow moving past it. We are interested to see all this because we are interested to find how much lift the aircraft has generated, how much drag is incurred when we fly such an aircraft. All these informations are very important for us. This is more like a flow visualization where you get to see which parts of the flow is attached and which parts are separating, so that gives you a lot of information about how much drag is produced by this body.

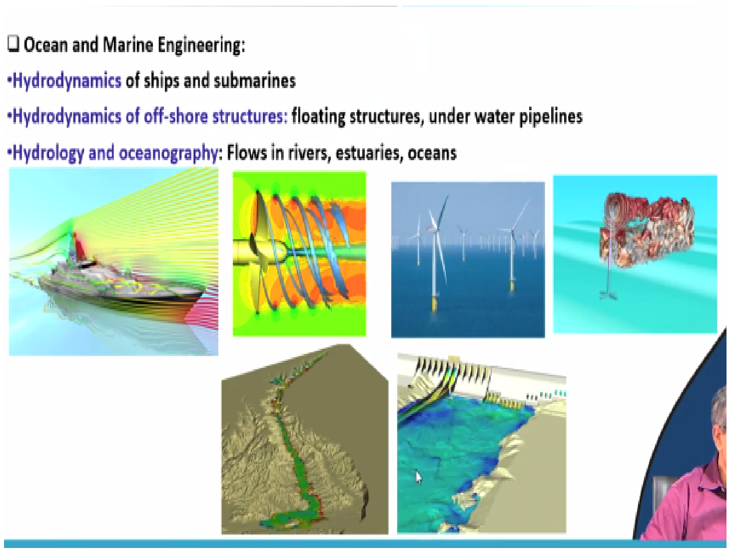
(Refer Slide Time: 09:12)



These aircrafts operate at much higher speeds than the ones we saw in the previous slides and they could be also doing a lot of complex maneuvers and you find them operating at very different orientations than the aircrafts which we travel by. So here, there are complex vortex structures, which could play a lot of role in helping the aircraft remain stable and perform its maneuvers.

This is a rocket in course of its upward movement from the earth towards the outer periphery of the atmosphere and beyond, and you can find very complex patterns formed around in a flow which shows compressibility effects. That means you can have shockwaves, you can have expansions and things like that. So, all these form the mechanical and aerospace applications of CFD.

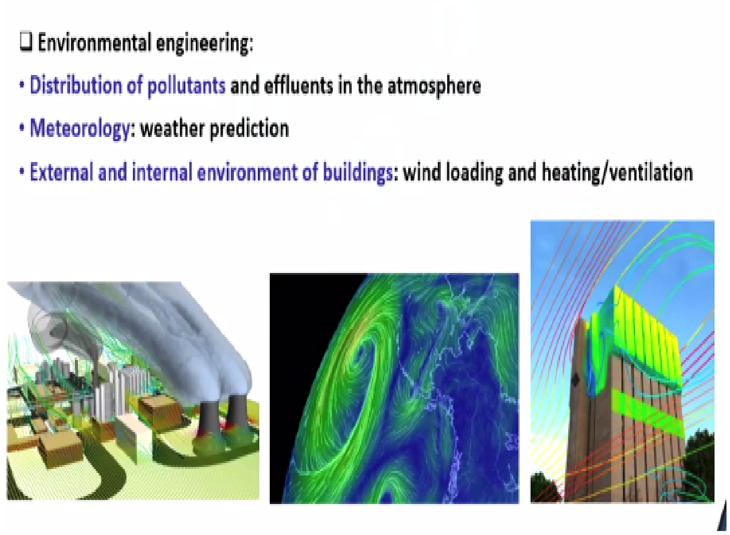
(Refer Slide Time: 10:20)



There could be numerous other applications. One that we are showing over here is related to ocean and marine applications, where you can be studying the CFD of ships and submarines. You may be looking at how the propellers are operating inside the seawater to create the propelling force for such a huge ship out here. You may be interested also to look at offshore structures like the ones here where you are operating wind turbines which are placed deep inside the sea.

Some of them may even be floating on the sea surface, where the sea is really very deep and then they may be moving with time because they are facing strong winds and then the CFD could really become complicated because you have to have moving surfaces accounted when you do your CFD calculations. Here, you are talking about rivers and estuaries or water being released from dams. So this is also a part of CFD.

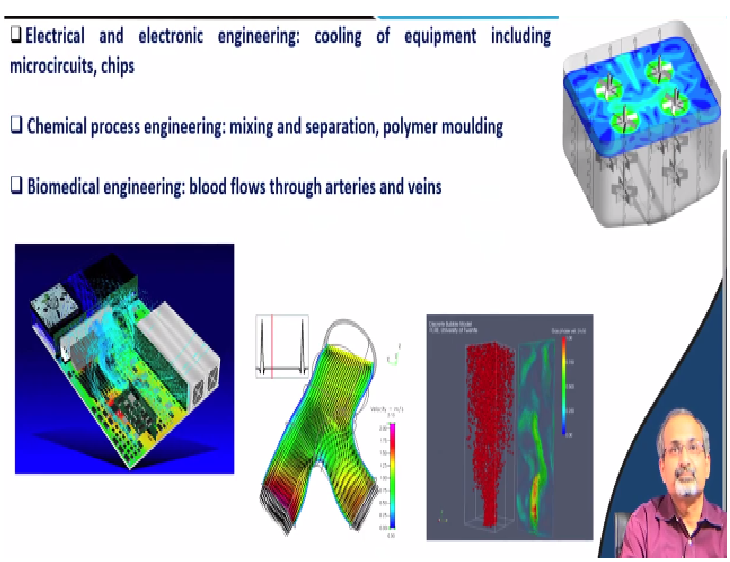
(Refer Slide Time: 11:35)



In environmental engineering, we are these days very much concerned about our environment, the pollution, the rising carbon dioxide content and other pollutant contents in our air. So we are extremely curious to know how well are we taking care of the effluent and pollutant dispersion. CFD plays a massive role in performing such simulations. We are concerned about weather in long time forecasts.

That means we are very well informed about major catastrophic behaviors of our weather. So CFD plays a big role there. In a given locality, we are concerned about buildings where we live, where we work. So, we are concerned about whether those buildings are strong enough to withstand wind loading, whether the heating and ventilation in such buildings are planned well, so that we get the desired comfort. CFD helps us to do that.

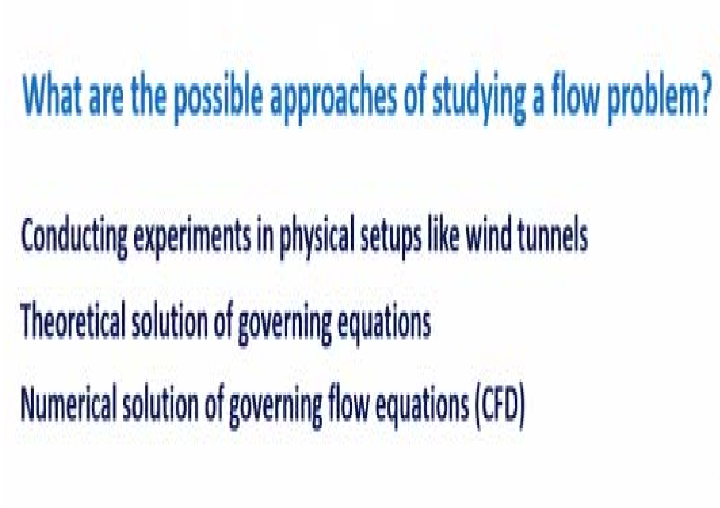
(Refer Slide Time: 12:42)



More applications to come. In the contemporary world, we have a lot of electronic gadgets all around where miniaturization has become a major part of the design paradigm shift and therefore you have a lot of heat generation within a very small space and then you have to have very effective means of dissipating the heat, CFD plays a major role there. It plays major roles in biomedical engineering to model flows through blood vessels, heart, and other components of the body.

It could be playing a role in chemical process industry and so on. So, there is a huge list of possible applications which you can look at, there is virtually no end to it.

(Refer Slide Time: 13:37)

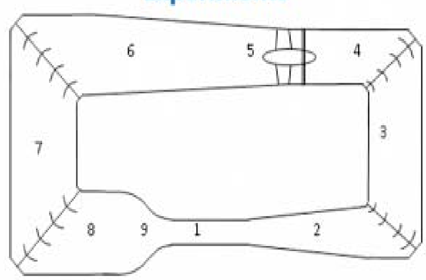


So, if you look at a given flow problem, is CFD the only way that is the question we need to pose ourselves. So, how we go about it is like this, we have a number of ways. We could be conducting experiments in wind tunnels, which are physical setups where you can create flow and you can study a given flow problem. That could be one way. You may be having theoretical solution for certain flow problems in which case you try to generate an analytical or a theoretical solution for the governing equations.

But that is more of an exception because most of the real-life problems unfortunately do not seem to be well addressed by most of these theoretical models, and then we have the possibility of numerical solution of governing flow equations, which is essentially CFD.

(Refer Slide Time: 14:36)

Experiments



Layout of a low speed closed circuit wind tunnel

- What you see is what you believe – often the most realistic way of solving a flow problem
- Wind tunnels, measuring instruments, model fabrication and instrumentation are costly
- Many quantities may be difficult to measure
- Simultaneous matching of similarity parameters may be difficult if not impossible
- Scale effects

So, if you look a little deeper into these 3 approaches, what do we really do when we talk about experiments in wind tunnels. So, here what we do is we have this big test facility which may actually run into several squares of meters as footprint and then there are different parts to this experimental equipment which we are calling as a wind tunnel. Now, this is nothing but a passage for airflow.

So, the boundary which you see on both sides are essentially made out of metal or wood or concrete and it confines the flow which is moving through it. Now, how does the flow occur.? You have a fan out here, which runs to create the flow and you have variable area passages which would accelerate or decelerate the flow suitably. You have the lowest cross section available here where we have marked it as 1 and that is where we put models and we test them when we run the wind tunnel.

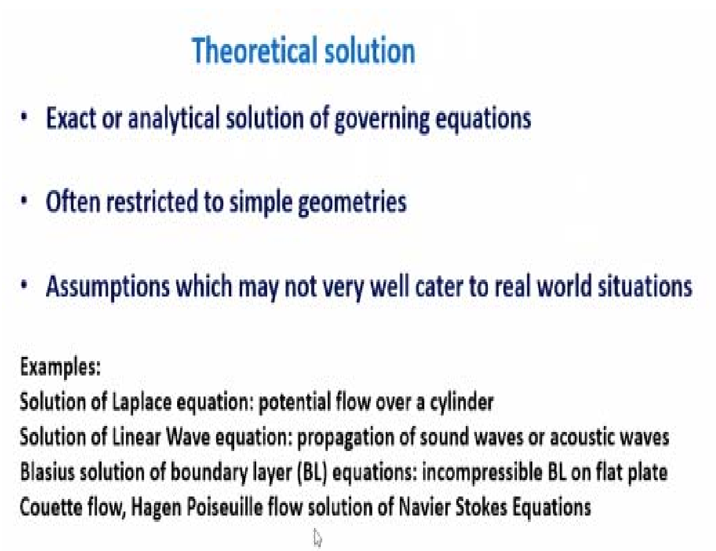
So, when you run the fan, there is a strong wind blowing through this section 1 and you may be actually having a small aircraft model or a car model or a chimney model sitting out in the test section. Then you can record velocity, pressure and other measurements can be undertaken based on what exactly we are looking for. Now, how well does this work? Well, we say what you see is what you believe and here in a physical experimental setup, you are able to see all that in reality.

It is not a simulation, it is not an animation, but reality, maybe in a scaled down sense. You may not be able to represent the entire aircraft or entire car model or the entire chimney in one is to one scale, but you can at least do it in a reduced scale and you get very close to reality. So, this could be one of the best ways of doing it, but wind tunnels could be very costly. Measuring instruments could also be very costly and therefore, the whole affair may not be affordable all the time.

Even running big tunnels could be very costly. Also in addition, when you perform experiments, there could be certain quantities which are not all that easy to measure and also experimentalists will tell you that there are still bigger issues when it comes to actually executing the experiments. So, one of them being that you may not be matching or able to match the similarity parameters simultaneously which you would like to.

To give an example, when you are testing an aircraft model in a wind tunnel test section, you may not be able to match the desired Mach number and Reynolds number simultaneously because of the scaling effects. Scaling effects could also affect the accuracy of measurements. You may not be able to maintain the same surface roughness on a model which you have scaled down to a very great extent because the surface finish that you can obtain on a very small model is not scaled down accordingly.

(Refer Slide Time: 18:04)



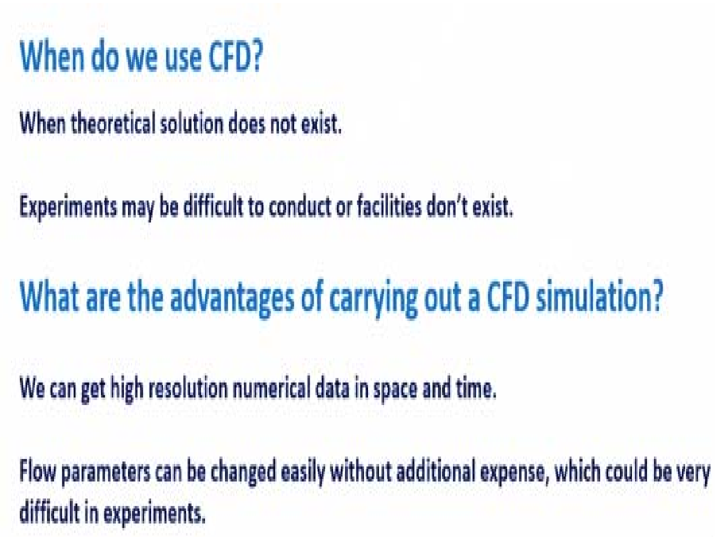
What about the theoretical solution? Well, you have an exact or analytical solution under the given constraints or assumptions. However, the constraints or the assumptions could be quite over restrictive at times and additionally you may be restricting to very simple geometries when you have these models working. So, these could be major limitations which often make theoretical solutions very restrictive in terms of application.

So, we have to make sure that the kind of problem that we are looking at or we are trying to solve whether it really fits in into this kind of a framework. We would certainly have the most accurate solution on earth, but we have to remember that it would be available most often for simple geometries and we have to understand exactly what the assumptions have been because those assumptions may be actually taking it far away from real world situations.

Some of the common examples that we can talk about in terms of theoretical solution in the fluid dynamics world are mentioned below in the slide as you can see in the form of examples. So, we talk about Laplace equation, which is a very important equation in fluid dynamics and even in other domains. It is important in heat transfer as well and if we are solving Laplace equation for fluid flows, we can think about applying it for potential flow over different geometries.

One geometry could be cylinder, it could be many other geometries that you can think of, let us say airfoil as well. So, for the cylinder at least we can actually generate a theoretical solution when we solve Laplace equation. When you look at linear wave equation, this solves the problem of propagation of sound waves or acoustic waves exactly. When we look at boundary layers which are forming on a flat plate that means the very thin viscous layer which forms on a flat plate when you expose it to a viscous flow, then you have an exact solution of the boundary layer which is credited to Blasius. You have several other flows for which you have solutions available. These solutions are from more complex system of equations, which we call as Navier Stokes equations. So when we apply certain assumptions, we are able to get solution to two very important solutions, one being Couette flow, the other is the Hagen Poiseuille flow. So, one applies to flow through a channel while the other applies to pipe flow.

(Refer Slide Time: 21:02)

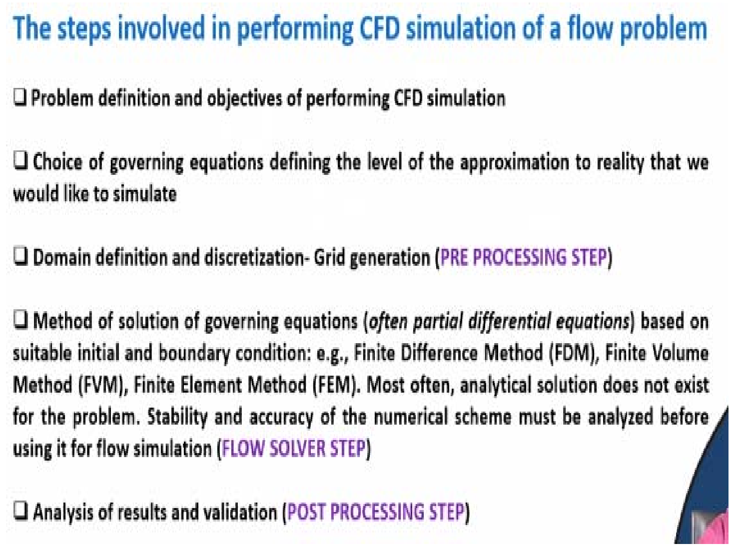


Now, coming to the point that having understood that there could be different approaches to solving a certain problem, when do we use CFD? The possible answer could be that we are running short of theoretical solution. So the theoretical solution does not exist for such a problem that could be one reason. The other reason could be that experiments may be difficult to conduct or facilities do not exist or experiments may be almost impossible to conduct at times.

So then you have enough reason why you should think about CFD. What are the advantages of carrying out a CFD simulation? Well, these are 2 important ones, there are many others. So let us say what we have here. We can get high resolution numerical data in space and time. What it essentially means is that through CFD when you are trying to solve a flow problem, which is dependent both on space and time, then by and large you can control that how much detail is required for that given problem, both in terms of space as well as time. That means, you can look deeper and deeper into space by zooming into smaller space intervals and deeper and deeper into time by zooming into smaller time intervals and CFD can ideally do that for you so that you finally have a very clear picture of the phenomena, both in terms of how it changed with space and how it changed with time.

Additionally, flow parameters can be changed easily without additional expense, which could be very difficult in experiments. For example, let us say you have an experimental facility where the maximum speed with which you can operate the facility is restricted by a certain value, so you would not be able to generate data anyway beyond that velocity. However, it is very simple most often to do it through a CFD exercise.

(Refer Slide Time: 23:12)



These are the main steps involved in performing CFD simulations of a flow problem. There are broadly 5 steps. We start by defining the problem and the objectives with which we are performing the CFD simulation. So, we have to be very careful in defining what exactly we are looking for. So that comes in this first step and a lot of the remaining steps are actually dependent on this. If you go to the next step, you see the choice of governing equations, defining the level of approximation to reality that we are actually looking for.

So we may or may not always look for very close approximation to reality because very close approximation would generally be very expensive. So, we have to essentially do a trade off here and accordingly the governing equations get defined. So, you may say that if you are doing a lot of trade off and you are compromising, then the governing equations in general will look simpler and vice versa.

Then comes the issue of domain definition and discretization. When we do CFD, we have to define a certain spatial domain over which we solve the problem, and if the problem is time dependent, we also have to decide how long we are going to actually follow the simulation. So, here we are primarily talking about the spatial part of the problem. So, if you have defined a domain for yourself, then in CFD we will find later that we have to actually break up that domain into bits and pieces, then apply the governing equations to those individual bits and generate a solution out of it. So, there comes the requirement for breaking down the domain into smaller bits, which we are calling as discretization. So, this process is called as grid generation. This forms a part of a major process which we call as the pre-processing step. Method of solution of governing equations: when we are doing computational fluid dynamics, we often end up handling partial differential equations and these equations can be solved based on certain suitable initial and boundary conditions.

They may be solved by numerous methods, but some of the more popular methods are mentioned here; finite difference method, finite volume method, finite element method and so on. Most often we do CFD because we do not have analytical solution, but when we do CFD, we will be essentially approximating the governing equations and we use numerical schemes to do that and we have to check for the stability and accuracy of the numerical scheme before we actually use them to generate the flow simulation.

So, this is a very major step which we call as the flow solver step. So, we are talking about solving the governing equations which is the heart of the activity. After we have solved the equations hopefully accurately, we try to analyze the results and if we are doing it for the first time and we do have some references with which we can compare the results, we try doing a step which we call as validation.

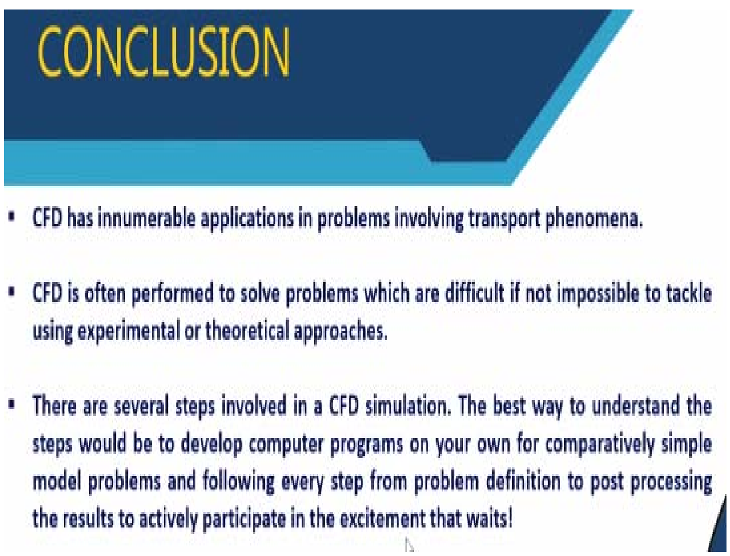
That means we really need to do cross check that how good did the simulation run? Is the result acceptable? Is it comparing well with results which are already reported elsewhere? So this is the validation step and it is a very important step of CFD. This is the step which actually gives us confidence about the results that we have generated. This is what is called as the post processing step and we need to remember there is more to it than just validation.

We will be doing more analysis on what we have generated and that could give us a lot of interpretation of the flow.

(Refer Slide Time: 27:43)

I am mentioning a few possible references here which could be handy for you. So we have a number of books recorded here, which can give you an initial introduction to computational fluid dynamics and also to fluid dynamics and aerodynamics in general.

(Refer Slide Time: 28:05)



So, we conclude saying that CFD has innumerable applications in problems involving transport phenomena in the contemporary world. It is often performed to solve problems which are difficult if not impossible to handle using experimental or theoretical approaches. There are several steps involved in a CFD simulation and we have to follow each of them meticulously, and here comes a very important point that when you are doing this course, you must feel enthused to develop computer programs.

This gives you some feel of what these steps are all about, so we need to pick up some comparatively simple model problems and follow all the steps so that we can actually solve the problems from scratch and then you can actively participate in the CFD exercise and this will be a lot of excitement I am sure. Thank you.