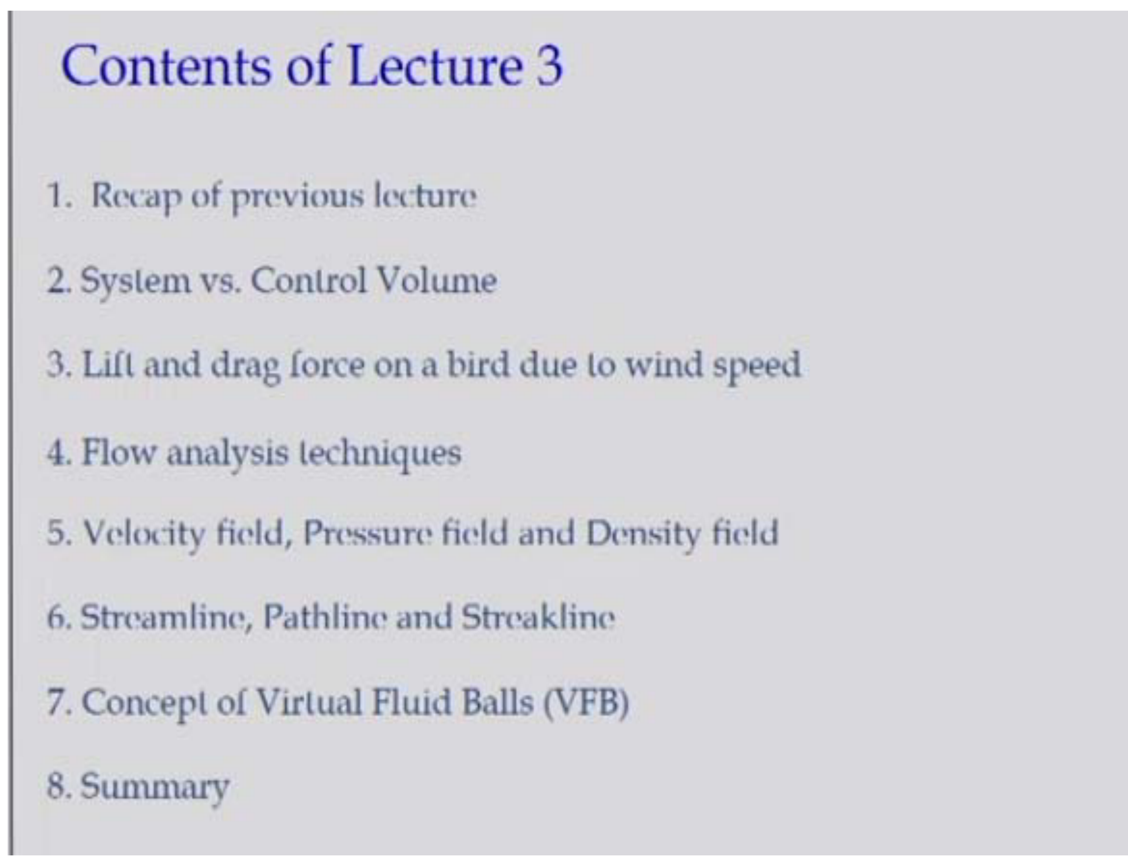
Fluid Mechanics

Prof. Subashisa Dutta
Department of Civil Engineering
Indian Institute of Technology-Guwahati

Lecture - 03
Fluid Flow Analysis

Welcome all of you for this third lectures on fluid mechanics. So today we will cover on flow analysis of very complex flow processes, how you can solve very complex flow processes. So that is what I will discuss today.

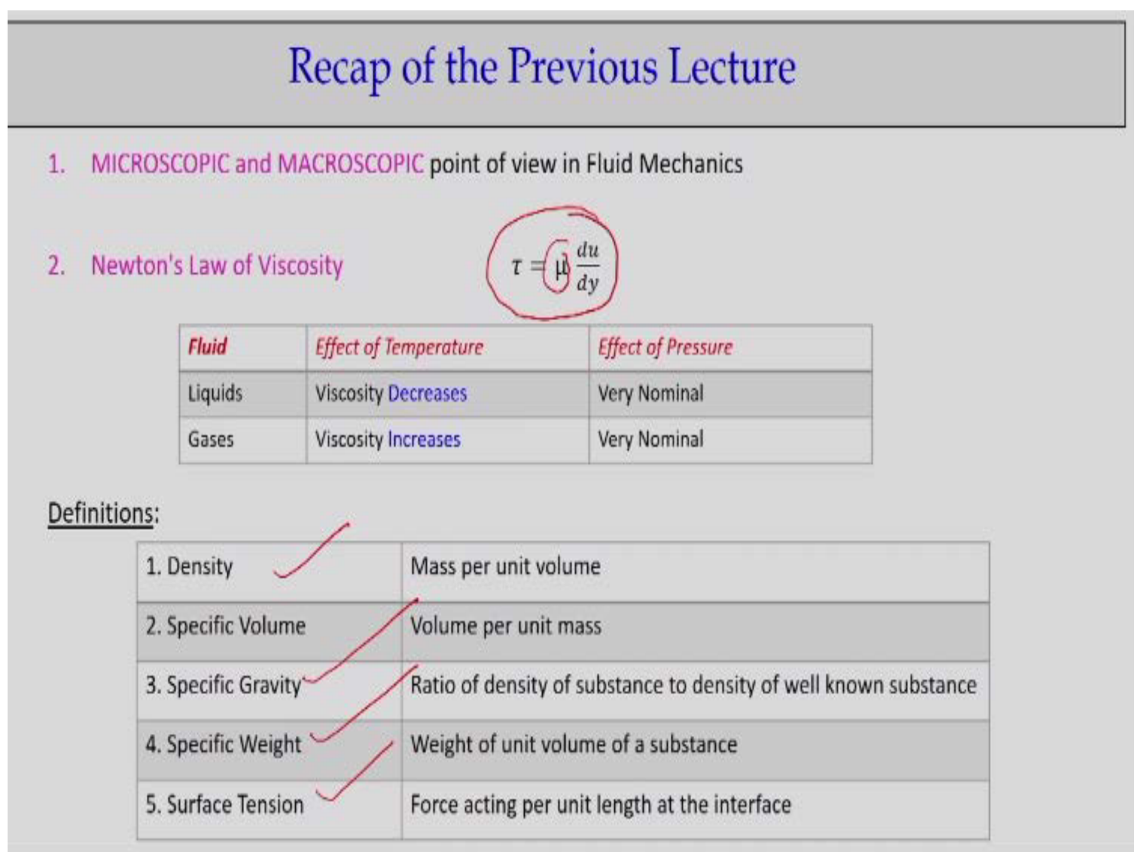
(Refer Slide Time: 00:55)



Considering that I will cover today systems and control volumes; what is the difference between system, what is difference between control volume. Then I will give a very interesting examples of a bird under the wind flow conditions. Then we will talk about what type of flow analysis techniques are available and how we solve very complex flow problems using these analysis techniques and then I will talk about velocity field, the pressure field, density field.

And after that we will talk about very interesting part is the streamline, the path line and the streak lines which is very much required for analyzing or visualizing a fluid flow problem. Then again, I will talk about this virtual fluid ball concept what we introduced in the first class. So same things how we can use it for very complex flow problems. Considering that let me look at what so far we have discussed.

(Refer Slide Time: 02:13)



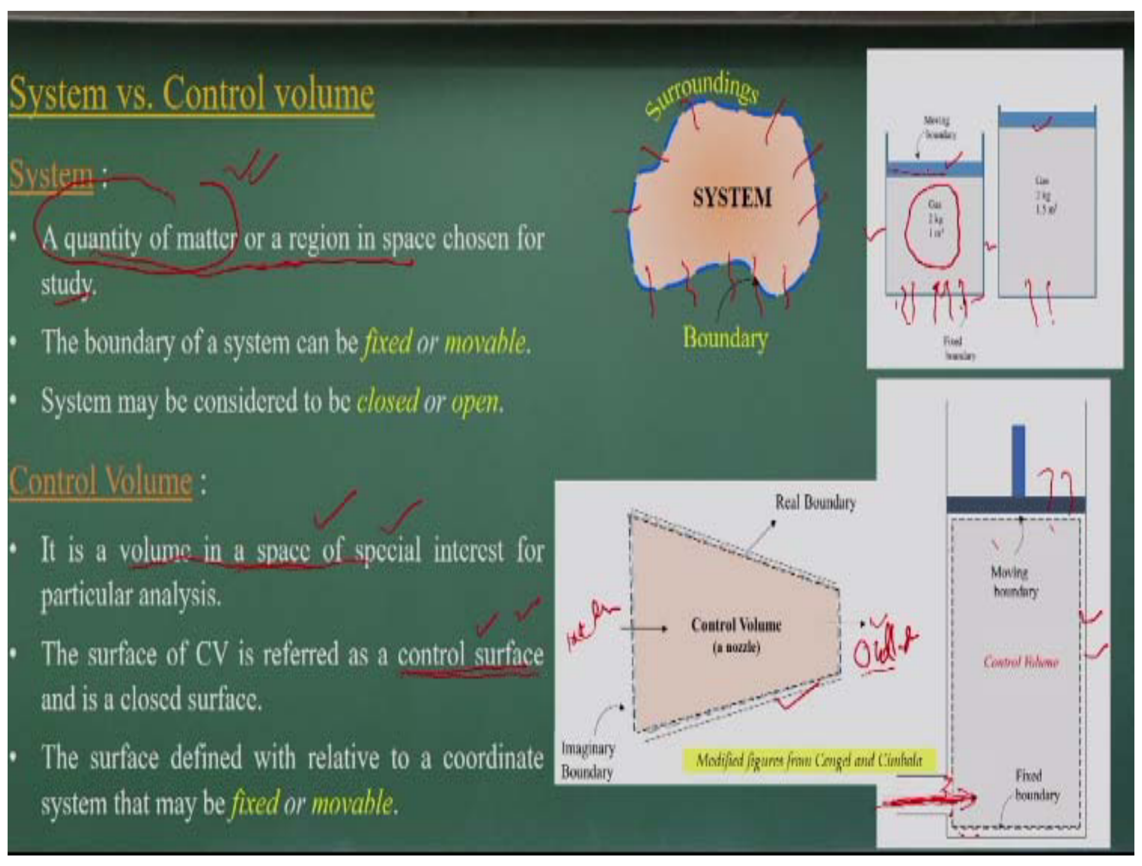
One is very basic equations of Newton's laws of viscosity which is establish a relationship between the shear stress and the shear strain rate and also we discussed about how these the μ the dynamic of viscosities depends upon the pressure and the temperatures,

$$\tau = \mu \frac{du}{dy}$$

This relationship is different for the fluid and for the gas part and after that we just know some of the basic definitions about density, specific gravity, specific weight and the surface tension.

With this knowledge, let us go to the next level that if I have a very complex problem, fluid flow problems, how do you solve that ones.

(Refer Slide Time: 03:08)



First let us talk what is the system, what is the control volume. The system is a quantity of matter or the region in a space chosen for the study. For example, I have considered a 2 kg of gas which is having 1 meter cube volumes. And if I heat this gas, if I give a temperature to this gas, then what will happen? This gas will be expanded.

So this is a system, that means we have a fixed amount of the mass of gas we consider it is a system and this system has a boundary and the surroundings. So the boundary in this case is the surface where the heat flux is coming into the gas, gas has expanded. Because of that the boundary at this stop is a moving boundary conditions whereas other directions at the fixed boundary conditions.

So the basically when you talk about the system we have the boundary we have some surroundings. Mostly when you talk about the systems we consider a fixed mass of the fluid. And how it interacts with the boundary with respect to the heat, mass, and momentum exchange through these boundaries. That is what is called the systems.

Here very clearly say that it considers a quantity of matters or the mass components as a system. But many of the times we cannot solve the problems within system approach which in generally follow in thermodynamics. But in case of the fluid flow problems, we go for a space defined by a particular volume, okay. Like for example, I have this problem. If you look it this is what my control volume.

This is the space what I have considered as a control volume and the fluid is coming from this sides and this piston is moving in this conditions. So this is what the control volume and there is the surface confined to this control volume is called the control surface. That is what the control surface. The control surface can be a fixed surface or can be a movable boundary conditions or these control surface here the mass or momentum flux entering to this control volume.

So we have a control volume the fixed regions or the movable regions on the space that what we consider it and it is confined by the surface we call the control surface and this control surface through these control surface the fluid mass momentum exchange mass or energy mass comes into the control volumes. But if you look it another case like you have the nozzles, you have the flow is coming from the left to the right and it has consider this control volume like this, okay.

So in this case, this is the inlet path and this is an outlet path. The flow is coming from the left side and it is going out the outsides. We have considered the boundary. One is the real boundary conditions another the imaginary boundary conditions. So you have considered a fixed the space in the fluid regions. Within that there may be a system of or a fixed mass of the fluid is make enter to this control volumes.

After certain times may again go out from these control surface. So we have a two approach. One is a system approach another is a control volume approach. The mostly in the fluid mechanics problems what we will solve it we will follow the control volume approach. That means we will define a regions defined by the surface that is your control surface.

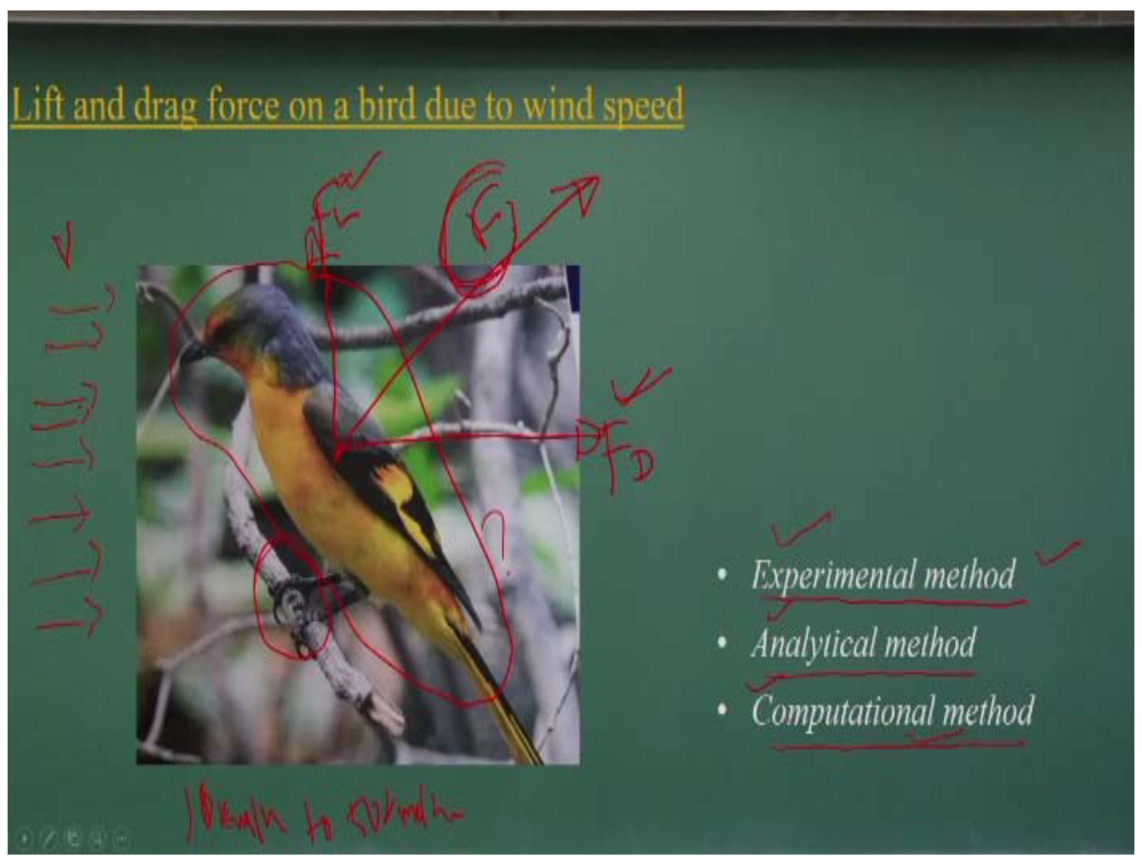
Through this control surface the fluid mass, the fluid momentum flux or the energy flux will come into this control volume. Also will be go out across this control surface as is the outlet conditions. So the mostly in fluid mechanics we follow the control volume approach, which is easy to solve as compared to the system approach where you have to consider a fixed mass of the fluid and you track over that which is very difficult when you have a very complex problem.

Whereas if you consider a control volume that means you consider a fixed regions of the space maybe in a fixed conditions or the movable conditions. That not a big problem. Here the easy things is that there will be a very defined surface, control surface in which the momentum flux, the energy flux will come into the control volumes. Also there will be a defined surface.

On that defined surface the mass flux, momentum flux, energy flux will go out. So the reasons we define it is the control volumes and the surface what you define is it the control surface. So it is easy to solve the fluid mechanics problem using the control volume approach.

So most of my lectures I will cover through these the control volume approach as compared to the system approach which mostly follow in the problems like what I am talking about heat flow and moving boundary conditions like a piston conditions which is most of the times in thermodynamics it is followed it but in case of the fluid flow, we consider the control volume approach to solve the problems.

(Refer Slide Time: 09:26)



Now, if you look it the next very interesting problems what I have to give a illustrations to you that if you look at this very beautiful bird sitting on a branch. If there is a wind movement is coming from this and this wind movements consider let me the this speed is increasing from 10 km/hr to 50 km/hr, okay. The speed of the wind is increasing from 10 km to 50 km/hr.

The question which comes it at which speed this bird cannot hold this branch. That means after that critical speed this bird has to fly from this place okay that is very interesting topic what you can understand it. That means what we are looking it that there is a fluid flow is coming from these sides is having a speed let be the V the speed is what is coming upon that.

Because of that here you are going to have a two force components okay. One will be the drag force and other will be the lift force and will have a resultant force what is occurring, because this fluid where it is passing through that that what will create a flow structure such a way that there will be a drag force there will be the lift force.

And you will have a resultant force what is occurring because this fluid which is passing through that, that what will create a flow structure such a way that there will be a drag force, there will be the lift force. Also the resultant force will react like that. So these resultant force when you cross it the strength of the holding of the bird on this at that time, bird has to fly from these places.

So we are looking it that at which speed the force will be coming such a way that, that amount of the force this bird cannot withstand with this holding. So this if you look at that is very interesting problems. But it is also the complex problem in the safe. If you look at this, the beautiful bird shape okay, it is very interesting, the geometry what you have.

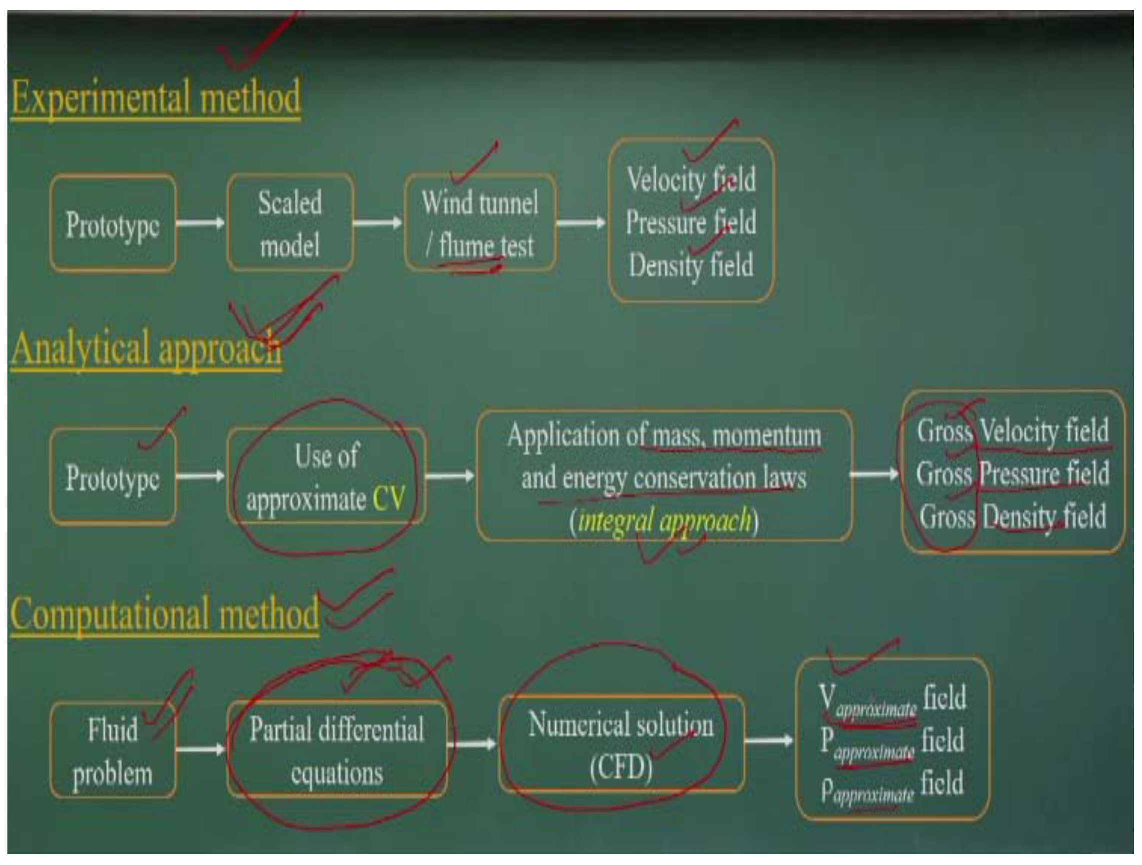
So also, as the wind flow is happening it that bird how it is responding which is a totally different aspect, we are not going to that details. So we are trying to look at how this force is going to happen it. So we can conduct the experimental way to compute it what will be the drag force, what will be the lift force, what will be the resultant forces or we can follow a analytical methods.

That means we can follow laws of conservations like mass conservation, momentum conservation, energy conservation, then we take a appropriate control volumes. Then you try to find out what could be the approximate the drag force and the lift force on this case. Or we go for very much the computational methods. The recently people has been using this the computational methods to solve these type of problems to find out what will be the drag and the lift forces.

So we have three ways to solve this problem. The experimental ways, analytical ways and also the computational ways. That means with help of the computers, by solving a set of nonlinear partial differential equations, we can find out what could be the pressure field, what could be the velocity field that what I will introduce you that. Based on that we can compute it, what will be the drag force, the lift force, and what will be the resultant force.

And at which force component this bird can withstand and beyond that it cannot. So that is what is the very interesting study can be looked upon. and

(Refer Slide Time: 13:38)



So similar way, that is what I am telling it, there are three ways to do the solve the any flow complex flow problems, like you can have experimental methods. That means you can have a prototype and make a scaled models. That means you can reduce the flow and the geometry of the problem in such a way that you do a scaled model.

Then you do a wind tunnel or the flume test to measure the velocity, pressure and the density. So once you measure the velocity and pressure and density that means you solve the problems. You know, how the flow is happening in terms of velocity, in terms of pressure, and same way the density. The second approach is analytical approach which mostly in the fluid mechanics books, we will cover with the analytical approach.

In that what do we do it we have a problem. We try to use a control volume, is a bigger control volume we try to use it. And we try to understand it where the mass flux is coming, the momentum flux is coming and which are the boundary there is no flux of mass, momentum, energy is passing through that. Equating applying this all mass, momentum, energy conservation with a integral approach we can find out the gross velocity, pressure and density.

Still I will talk about the gross. It is not the very detail distributions of the velocity the distribution on the pressure or the density we will get it. In a very average type of conditions what we can predict it as a gross characteristic what will go through this analytical method. One is experimental methods. The second is the analytical methods. Third is which is the last one of two decades is very famous is the computational fluid dynamics.

In which what we do it any of the fluid problems okay, we define through these mass conservations and momentum, energy conservation into a set of partial differential equations. This most often is a nonlinear partial differential equations and both equations we try to solve the numerically. As we solve this numerically, we get the velocity pressure and the density distribution, but these are all approximate solution.

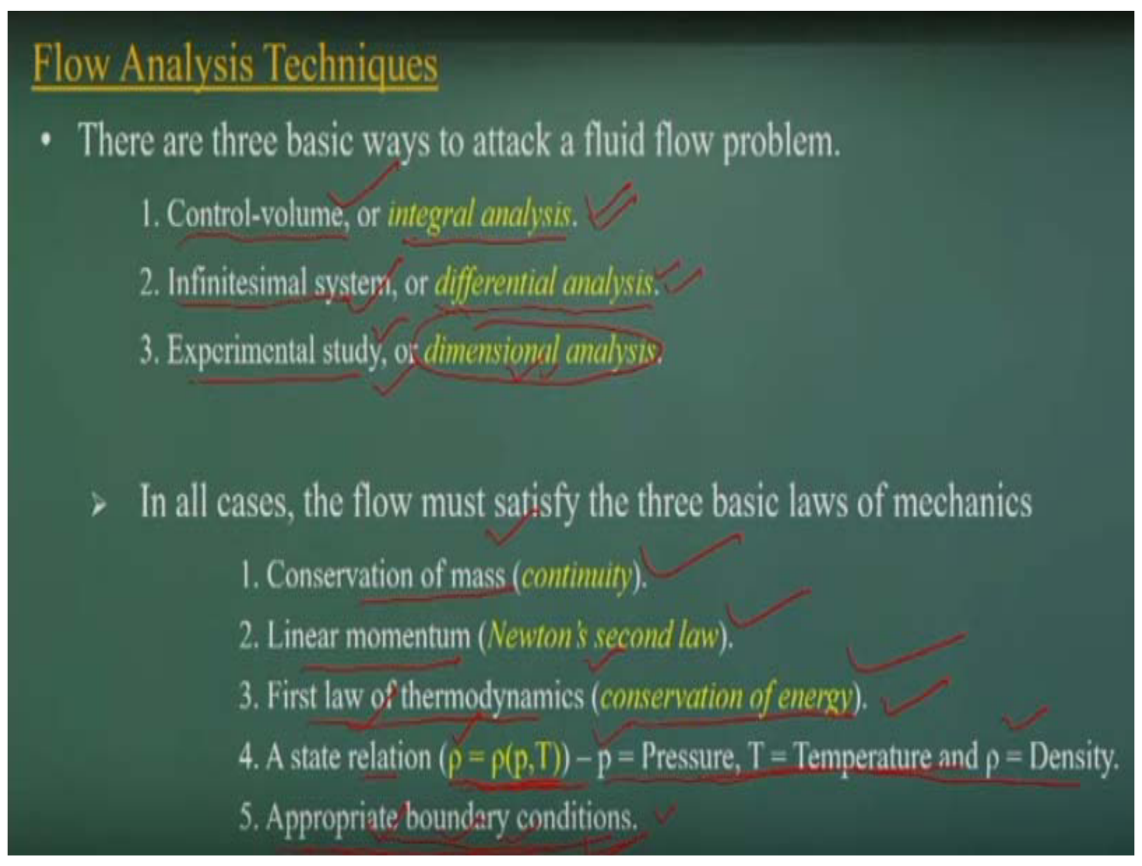
It is not the true solutions, because they are the numerical solution is approximation solutions to this numerical partial differential equation. So we get approximate solutions of the velocity field, the pressure field and the density field. So that in summary, I can say that we have three ways to solve the problems. One is conducting experiments to find out the velocity, pressure, and density field.

Second one is that we use appropriate control volume, apply basic conservation equations like mass conservations, momentum conservations, and energy conservation equation. Then we try to get it what is the gross velocity distributions, the pressure distributions and the density distribution. These are gross level. It is not gives a very detailed like the we get it the wind tunnel or the numerical methods.

In a computational method as you know it now we have a lot of supercomputers, we can solve many of the complex problems, fluid flow problems, that is what we do it any of the fluid flow problems we convert to a set of partial differential equations by applying same basic principle of mass, momentum and energy. But here these the control volumes are the unit what we consider that is what is infinitely small.

And then we try to solve that equation using a numerical method. And then we get this approximate values of velocity, pressure, and the density field. So we have a three ways to solve any complex fluid flow problems experimentally, analytically and computational way.

(Refer Slide Time: 18:09)



Now if you let me summarize that way, there are three basic ways to solve the problems. One is a bigger control volume where we have an integral analysis or the analytical methods ways. You take a smaller control volume, which is very close to infinitely small. Then you get a set of a differential equation problem. So that is the reason we call differential analysis.

Then the experimental study, as I said that we need to have a scaled model. So we will discuss more in the latter half of my lecture series that there is a technique to how to scale the models from the prototype to scaled models. How to get different geometry scale, flow scale all the similarity concept what we will discuss in later on. So that way you have control volume, infinite small systems.

That means analytical and you have a computational methods techniques, then experimental methods. And as I already said that, these three basic equations always need to be applied for any fluid flow problems. They are conservations of mass, the linear momentums or the angular momentums, the first law of the thermodynamics that is conservations of energy.

But apart from these equations we adopt a state relationship. That means a relationship between a density and the pressure and temperature, the ideal gas laws. So these type of the relationship between one variable to other the independent variable like pressure and temperatures, we can establish that relationship and that relationship is called the state relationship.

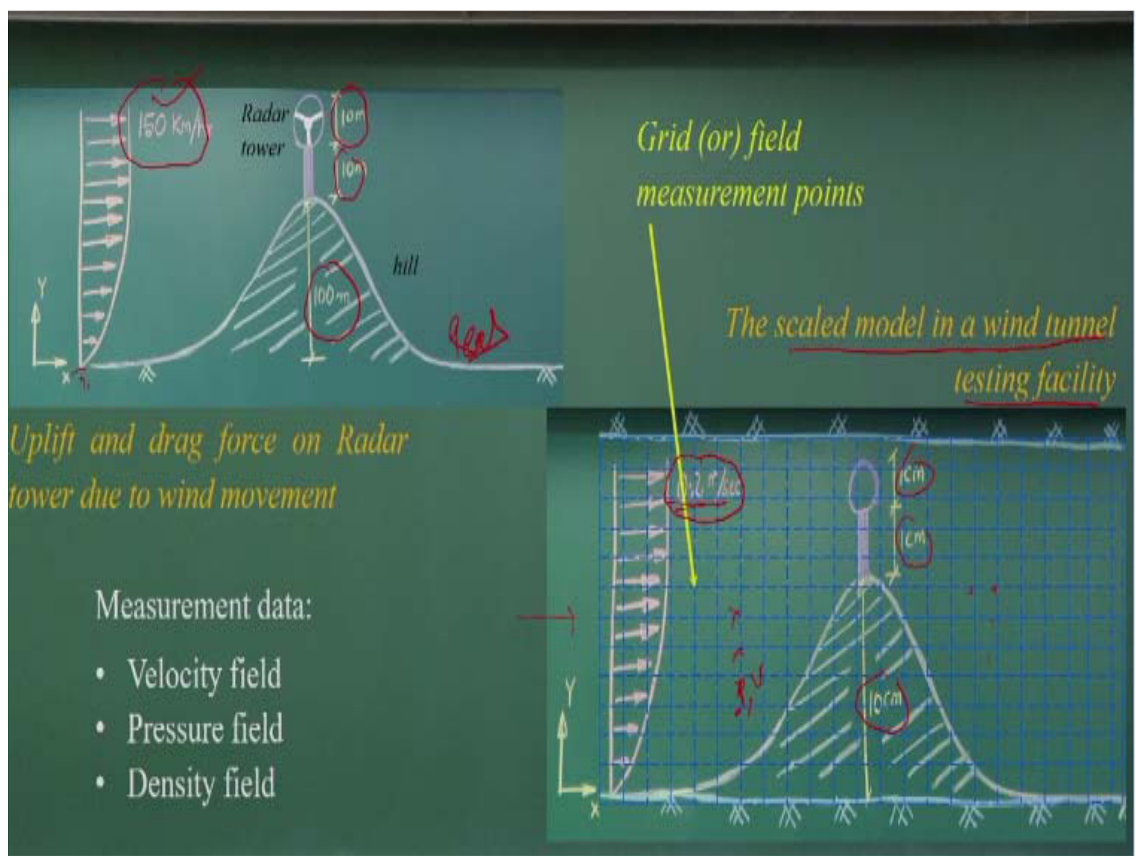
And at the last one what I can say that not only know this what type of flow problems also we should have a very good understanding how to define the boundary conditions. That means, you have to give a appropriate boundary conditions to solve the problems.

So the flow analysis techniques what is available to us with a very basic conservation equation as we apply for solid mechanics here also we follow conservation of mass, momentum, linear momentum equation, energy conservation equations. Apart from that, we look for appropriate boundary conditions and the state relationship to solve this problem.

So this is what the basic strategy to solve any fluid flow problems and a fluid specialist has to have a confidence or knowledge on how to define the boundary conditions, what type of equations to apply it and which condition he has to go for a control volume approach or the differential analysis or the experimental analysis that all it depends upon a knowledge on the fluid mechanics.

So that is my idea is that we should have a very in depth knowledge of the fluid mechanics then we can analyze a complex fluid flow problems taking appropriate analysis techniques. Now let us come to a very interesting examples here, we have given it here.

(Refer Slide Time: 21:44)



That they let be there is a weather radar is there. You know it nowadays weather radars are there to measure the rainfall, the wind velocities and all. That is what is a on the hilltop and it has a stand of 10 meters and the weather radar tower is about 10 meters. And the wind is moving at a speed of 150 km/hr and the velocity is 0 at this point.

If you have that conditions the questions is coming to design this radar systems we need to find out what could be the maximum uplift force and the drag force of this radar systems when you the wind speed is close to 150 km/hr. If that is the problems, for these type of complex problems we do not have any analytical solutions. So what we go for? We go for a scaled model in a wind tunnel, okay?

So that means in a wind tunnel, we set up the scaled models. As I say the scaled models means we reduce the dimension, we reduce the flow velocities or the densities. How to do that we will discuss that in a dimensional analysis chapters, but lets us you understand it that these are the real problems.

And we reduce to a scale models which is if you look it here is 100 meters here is 10 centimeters, the 1 centimeters, 1 centimeter. So we scale the models okay. We reduce the dimensions. Similar way we have reduced the velocity and this is the facilities is there in the wind tunnels and we have the velocity flow is going like this and at each point of this grid we will measure the pressure, velocity components, okay?

So if you look it that, so we have the wind tunnels and we will fix up these the scale models of the hills, the radar systems as equivalent to a spherical body and connecting part and we generate the flow systems which is you will have the flow distributions like this and with having a velocity 0.52 meter per second and each grid point will measure it what is the pressure, what is the velocity, what is the density.

(Refer Slide Time: 24:28)

For example, if you look it that point, like the wind tunnel facilities what you have at the IIT Guwahati in the Department of Mechanical Energy. See if you can look at this wind tunnel the setups okay there is an inflow, this will be the outflow and this is what the test sections what is there and these are all the recording systems for velocity measurement and the pressure measurements.

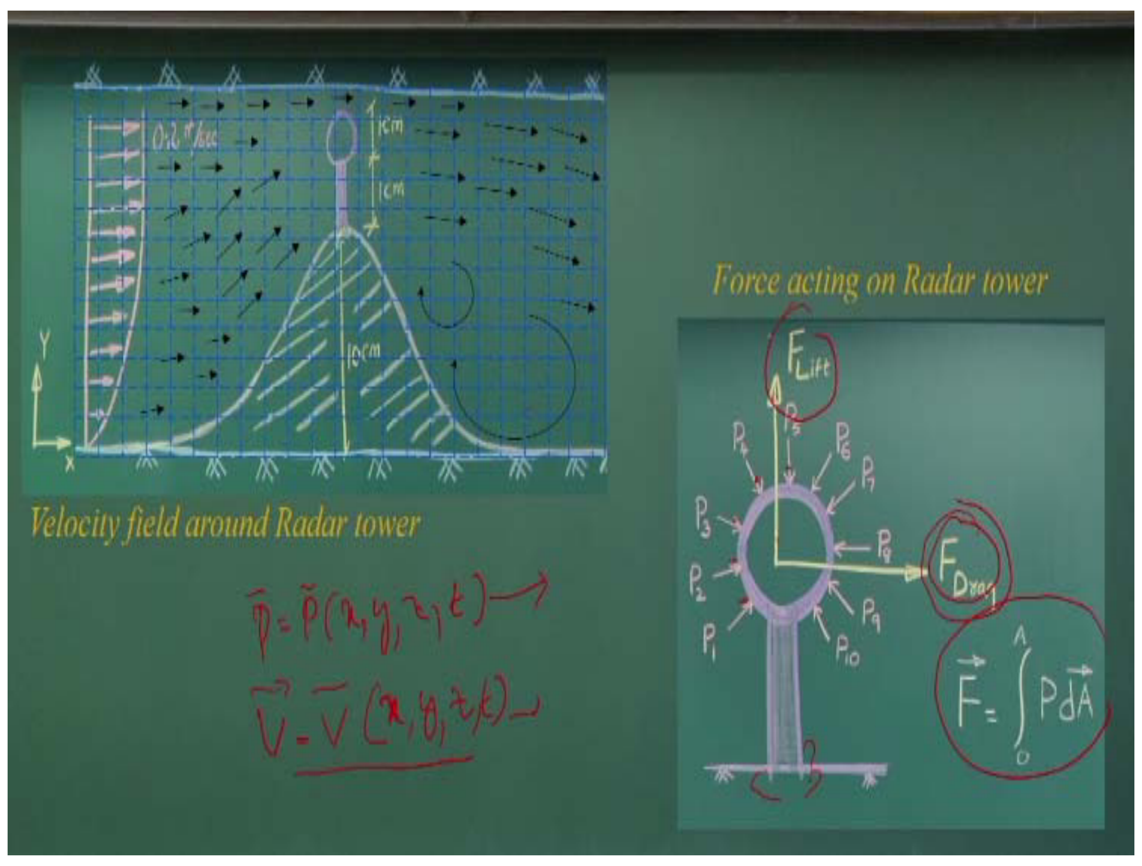
So if you can look at these type of wind tunnels they are most of the advanced fluid mechanics labs this type of wind tunnel systems are there where we generate the wind movement through a test sections like this. If you look it this is the or the test sections okay. In the test section we generate the wind flow. To measure the velocity, we have a Pitot static tubes. I will discuss more detail what is it a Pitot static tube in the later on.

So you can now know it there are the instruments which can measure the velocity, not only this one direction, it can measure the three dimensional velocity component. That means it can measure the u, v, and the w components, the three velocity distributions component can measure it. And similar way we can have a manometer. I will discuss what is a manometer and all the things later on.

That the instrument can use to measure the pressure. So we can measure the velocity, we can measure the pressure and this is the wind tunnels where you can the variable the speed, the speed of the winds, which will pass through these the test sections. Then we can measure the velocity and the pressure. Similar we can have also measure the density.

So if you look it this way, that means what I am talking about that I have a scale models put into the wind tunnel facilities. Then at each point we are measuring the pressure and the velocity and the density. It is the measurement, the physical measurement of pressure and velocity and the temp and the density.

(Refer Slide Time: 26:41)



Now if you look it for example, we got for each grid point this type of velocity factors. It has the directions; it has a magnitude. So we get the velocity at each points.

So interestingly you can see that there will be the high velocity zones, there will be a low velocity zones and there could be a vertex formation. But that is not our interest now. We just measure the at each grid point the velocity factors, velocity magnitudes.

So that means we know u, v, w component, the scalar component of velocity we know it. That is what is called the velocity field. Now you can understand it if I take a more number of grid points the velocity field will have a more accuracy. So if I take a less number of the grid point for the measurements, I will have a less accuracy in defining the velocity field because the sampling points are less.

So that depending upon the flow problems, you can decide that how much the spacing, how much locations you should collect the velocity or the pressure or the density content and once you know these pressure components that means I know it approximately how the pressure is varying with the space and the time. I know it how it varies because these are the measurement values.

The similar way, if I have vectors, this is what I measure it the vector which is the sample point vectors, which have varying it with x and the t. So this is what called the pressure field and the velocity field. Now if you look it that my prime objective as an engineer is that I have to design what could be the lift force and the drag force because of 150 km/hr the wind is blowing over these tower.

What we can do it one we can measure the pressure. That means you knew the pressure point P_1, P_2, P_4, P_5 like this. You just integrate that places you can find out what will be the drag force and the lift force components.

$$\vec{F} = \int P d\vec{A}$$

So this drag and lift force can be used to design this the civil engineering design the structural engineering designs of this tower.

The mounting staff or the foundations will design such a way that whenever when you have a 150 km/hr the winds pass over this tower system, these structures would be in safe, okay. So that is what to make it that we should know exactly what is the drag force and the lift force is happening it.

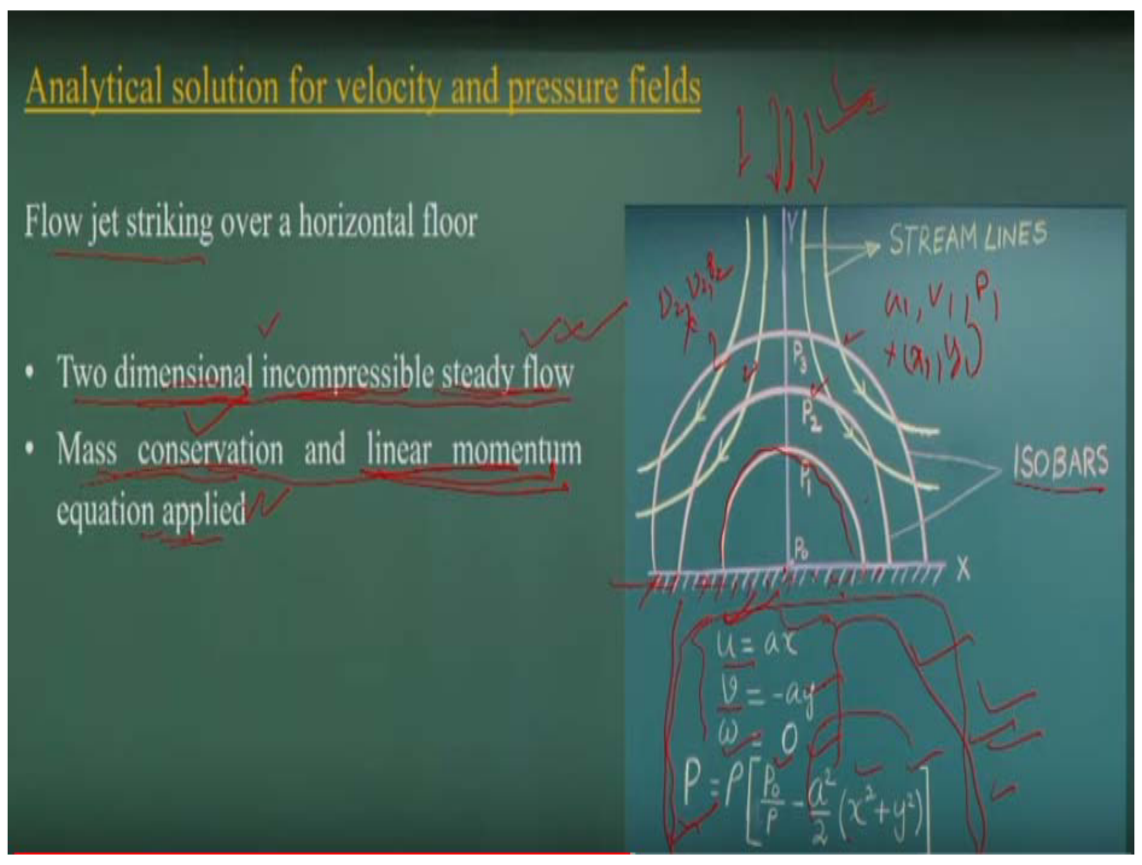
So with this example, if you can know it, that with measurements with conducting the experiments, we can compute it the velocity field and the pressure field and knowing this pressure field and velocity field we can compute it, what will be the gross uplift force is going to act it and the drag force and these two force can be used to design these structures for wind speed of 150 km/hr.

So this is what is experimental way we can conduct it. But you can always think it that the conducting this type of experiment is always expensive and for that we need to have a wind tunnel facility to solve these problems. Today's you know it there are big wind tunnel facilities are there. They full prototype of truck they simulate it.

So that is what am I pointing that the facilities nowadays the wind tunnel facility what is there in all over the world, the automobile engineers they use the full truck not the scaled models, they conduct the wind tunnel test to know it what will be the drag force, the lift force for different flow conditions like different velocity, different friction drag components they try to do solve that type of problems the very complex fluid flow problems with full scale wind tunnel lab facility.

So it is possible today's it is not that difficult that to conduct the wind tunnel test, but it is expensive as compared to the other methods what we are going to discuss it.

(Refer Slide Time: 31:41)



Second one you know it we are looking for a analytical solutions. That means the fluid flow problems we are more or less very simplified cases like for examples, we have a flow jet, that means flow velocity is coming from this striking on the horizontal floor here. The flow jet is coming and striking over a horizontal floor. And our interest to know it what will be the pressure distributions, what will be the velocity distributions.

These problems we can identify it is a two dimensional problems we can make it because z direction it does not have a much component. So incompressible flow and we can simplify to a steady flow. So we have simplified the flow now okay as a two dimensional incompressible and steady flow. Because of that, we can use this the mass conservations and linear momentum equations.

Okay, I am not going more detailed. Anyway, we will discuss that how to apply the mass conservations and linear momentum equations here. Considering that if I do that, I will get a solutions of this. These are integral solutions of the scalar velocity component of u, the v as a function of ax and ay and the pressure will be function of x and y; a is constant and P_0 is a pressure at this point at the where we have the origin.

The pressure at that point which called the stagnation pressure. So if I know these equations, which satisfy this conservation equations and the linear momentum equations and the boundary conditions at this point as well as this boundary conditions here, then this is what my solutions of the equations which is the analytical solutions of equations.

That means, if I take any point here, if I have x_1 and y_1 is the coordinate I just substituting this x and y_1 I can get the u_1 component v_1 component and the P_1 component. Here the u_2 component v_2 component and P_2 component. So here what we have drawn it as the water jet is coming it and tracking on the horizontal floors. As you know it that this flow is a symmetric problem.

So exactly at the center the flow velocity will be zero. That is what you can say that if you substitute $x \: and \: y = 0$ v and v component will be zero. So that condition is satisfied and the pressure if you look it that it will be varied with $x^2 + y^2$ is a equations of a circles okay it is more or less the equations of a circle.

That is the reason you will have a unit for the pressure, the constant pressures line like this, the concentric circles like that, what will be half concentric circles. So this is what called isobar. The line of equal pressures. So you will get the equal pressure line. And if you know the u, v and that you can draw the flow stream lines. That means flow will come like this okay the pressure diagrams like that.