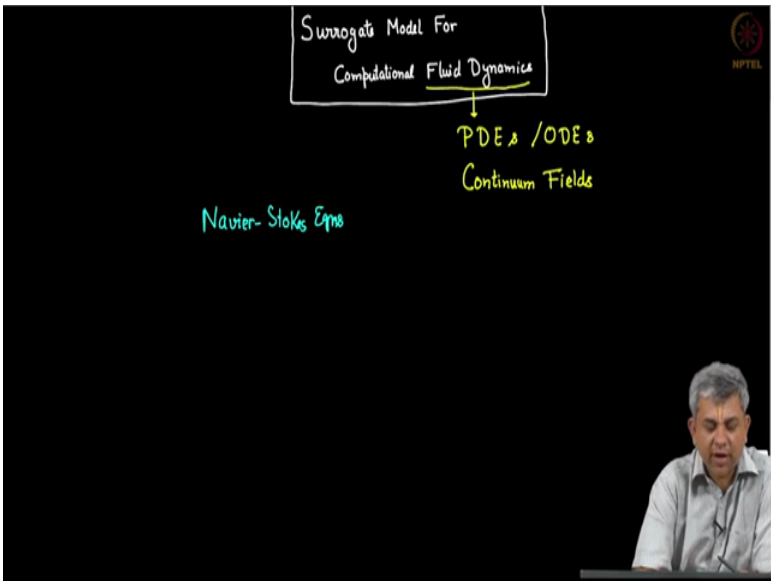
Machine Learning for Engineering and Science Applications
Professor Dr. Balaji Srinivasan
Department of Mechanical Engineering
Indian Institute of Technology, Madras
Application 2 description Computational Fluid Dynamics

(Refer Slide Time: 0:14)



Welcome back, in this video we will be looking at the second application, the previous application was a very straight forward one, we had three inputs and we simply had one single output, the inputs were heterogeneous there and we had a simple output which we wanted to predict the temperature at a particular point in a certain body and there we saw that artificial neural networks actually perform really well.

We are going to now look at a slightly more complex example right now this is one of those applications which has come up only in the last couple of years. So, as usual as I had said earlier, I will simply be talking about the application right now in this video and I will let you think about how you could go up about doing this and in the next video we will actually look at how people have actually solved it in recent years.

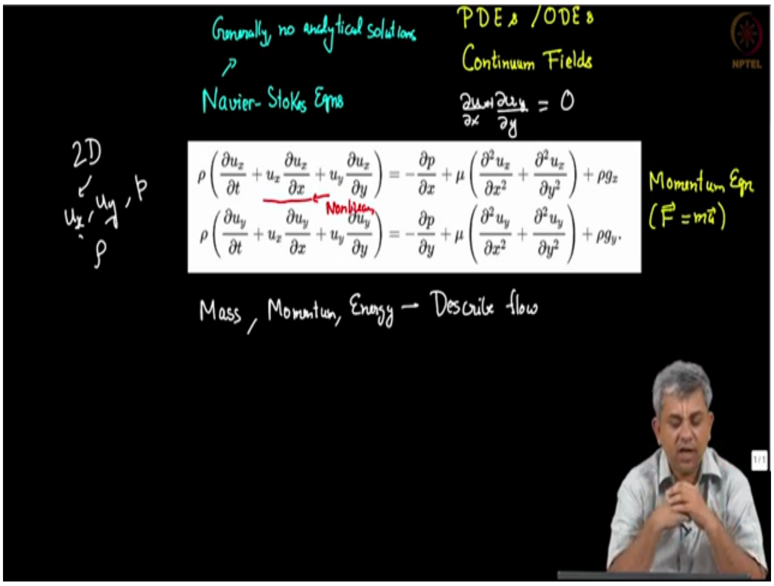
So the problem here is that of a surrogate model for what is called computational fluid dynamics, computational fluid dynamics happens to be actually the area that I work in most extensively. Now what it is as the name suggests it is fluid dynamics done computationally.

So fluid dynamic simply is the prediction of fluid flow, those of you who are not in fluid dynamics can still appreciate the kind of example that I am giving in case you are doing any problem at all in engineering or science, you will typically find some partial differential equations or in some cases you might find ordinary differential equations, these tend to happen whenever you are dealing with continuum fields, continuum fields means we assume that we are dealing with continuous quantities even though we know atoms exist, we typically model a fluid as if it is a continuous thing and as a solid as if it is a continuous thing and all this belong to the general field of continuum mechanics.

Now the ideas that I am discussing can be applied to electromagnetics, the Maxwell's equations or fluid dynamics what are called the Navier–Stokes Equation or solid mechanics etc you can apply it to practically anything including let us say relativistic equations, wherever you have any governing PDE or ODE the vertical idea that I am going to discuss will actually be pretty useful.

Okay, so the problem is as follows, you have the governing partial differential equations these are called the Navier–Stokes Equations let me shows those equations to you just so that you get up flavour of what they look like.

(Refer Slide Time: 2:58)



So these are the Navier–Stokes Equations, they represents the ones that I have shown here represent what is known as Momentum Equation and they represent the fact that F equal to ma that is Newton's second law is applicable to a fluid also so that is where you get this, these terms that you see here as you can imagine when I want to describe a fluid flow let us say it is around an aircraft or around a car or inside an air conditioner wherever it is all of it has three properties that we know are satisfied we know that mass is conserved, we know that momentum obeys Newton's second law and we know that energy is conserved.

So these three put together described fluid flow, now even though they describe fluid flow you can see that the equations as they occur here are fairly complex.

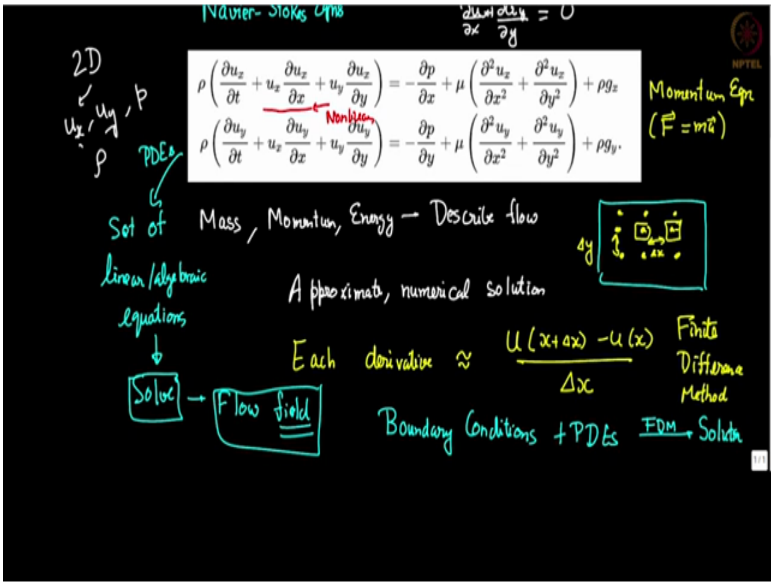
Now very quickly if you have a 2 dimensional flow that is let us say the flow over you are just looking at a plane channel where you are looking at this flow, so if we look at a 2 dimensional flow it will have three quantities that we would like to describe u x that is the velocity component in the x direction at a particular point, u y the velocity component in the y direction at a particular point and pressure, so pressure at a particular point.

Of course you would have density also, but density in case it is what we call incompressible which is what I am going to assume for this video and the next as long as it is incompressible density is a constant, okay. So you have two equations here for two of these variables, you also have a third equation this is the mass equation this is written in this way, once again none of you have to know this in order for the exam or for you to successfully do the (())(4:59) this is just to appreciate what we are going to do in this video and the next.

Okay, so as you see this you will see that the equation is actually fairly complex in fact this term is somewhat nonlinear and with fluid mechanics with Navier–Stokes we know that there are generally no analytical solutions, so you cannot solve them like the analytical solutions that you would have found in class, etc people in CFD know this very well, in fact this is a very big problem.

Now how do we solve this in general? If we cannot have analytical solutions, how do we actually solve the fluid flow equations? In fact the planes that you travel in let us say a Boeing, etc all are designed on the basis of solutions to these very equations. So this is actually an advanced art and science right now to actually solve these equations and being able to predict what happens in a flow outside a body etc this is really how we design things.

(Refer Slide Time: 6:04)



Okay, so suppose you want to predict this flow, what you do is apply an approximate numerical solution you cannot find exact solution, so you find out an approximate numerical solution and the way you do it is let us say each derivative for example the derivative here del u, del x you approximate it as u at x plus delta x minus u at x divided by delta x. Of course I am giving you a very simplified and simplistic example of how we do this.

So the way we do it is instead of having this continuous field which is what you will get, in case you have an analytical solution, what you will do is, you will say I want my solution at some finite points, tell me the solution at let us say nine nine of these points I just want these nine points. So what you will say is, well I will approximate the derivative here as the value here minus the value here by delta x or the value here minus the value here by 2 delta x, where delta x is this distance.

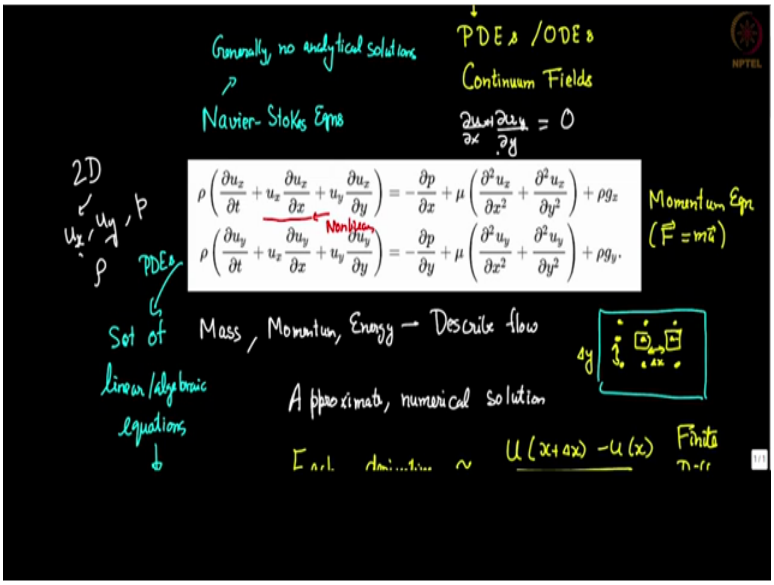
Similarly whenever you have a y derivative, you can approximate this by delta y, so on and so forth. Okay, so this is called the Finite Difference Method this was one of the first methods that was tried about (a thousand) about a hundred years ago, okay. Now we have come to much more sophisticated schemes, but the basic idea still remains the same, what happens is this set of PDEs become a set of linear or algebraic equations, okay.

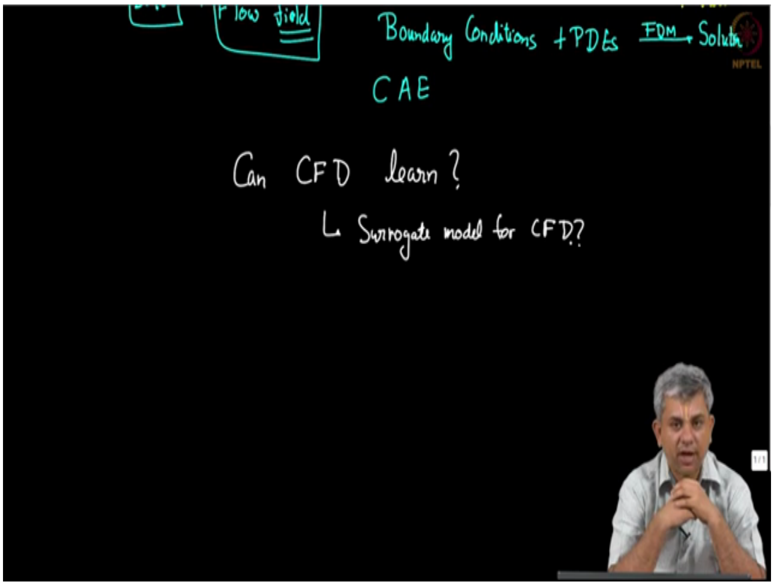
So each PDE is converted to a bunch of linear equations and then we know how to solve linear equations, we know how to solve combinations of linear equations or even combinations of algebraic equations and you get the solution. So this is how you get what is known as a Flow Field, so please underline the word field, field means you actually have a distribution of the same quantity everywhere, okay.

So this is our aim, our aim is to start from the equations and I will tell you the specific circumstance that I am in, the circumstance is usually given by Boundary Conditions that is obviously even though it is the same equations the flow past and aircraft is going to look different from the flow past a cycle it is going to different look different from the flow past a man, okay and what decides it? What the boundary of the object is like, so these are called boundary conditions, okay.

So PDEs plus boundary conditions passed through the finite difference method give us the solution. So this is the current situation.

(Refer Slide Time: 9:38)





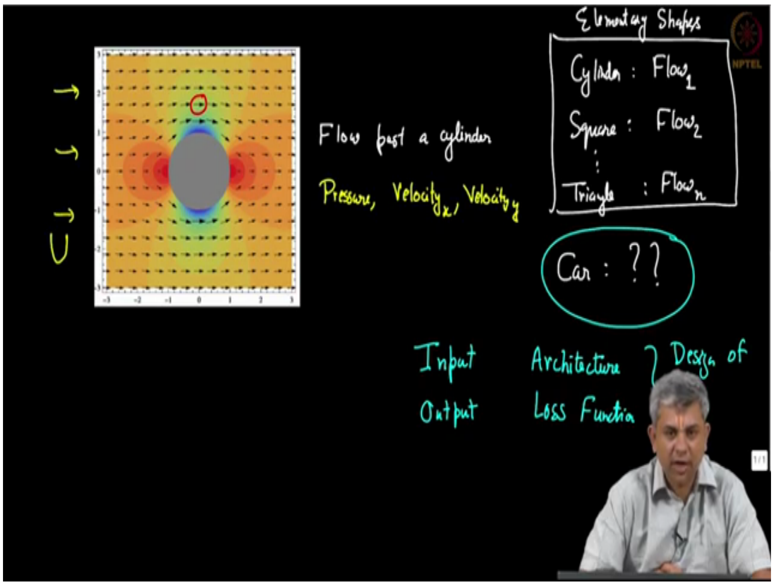
Now the problem with this is for complex bodies it can take you a very long time to actually solve these equations. So each time for example let us say I am doing a design of an aircraft or design of the shape of a car of a racing car, each time I make a small change in the shape of the body, I will have to go through a lot of computational time in order to get one solution.

In fact even in India at other places we know that this kind of computation called CAE basically you can find out lots of approximate solutions to problems that are facing you whether they are fluid mechanics or whether they are solid mechanics, we can solve these problems through the computational means and we know that even in India we spend a lot of money, you know big companies like Ford Chrysler, etc they spend a lot of money in just doing this that is because in each design cycle you have to go through many computations.

So here is the question that I want to race, the question is can CFD learn? What is meant by can CFD learn? What I mean is suppose I have done hundreds of computations in the past. So if I have done these hundreds of computations the hundred first computation should I be solving the same equations again right from scratch or can I somehow use my data from previous simulations and find out a simpler model that will run faster.

In other words, can I find a surrogate model? Can I find a surrogate model for CFD? Now why would this be useful? That will be useful because the example that I gave you when I want to do computations of any design of a let us say a car, instead of running the full CFD model, suppose I could run a simple surrogate model which will run much faster than I could get my solutions much faster, okay.

(Refer Slide Time: 12:04)



So before I leave you with this question let me show you a quick example of what kind of solutions we are expecting from this surrogate model? What you see here is what is called flow past a cylinder and what is flow past a cylinder? As you can see you have some fluid coming from here at a certain speed let us say capital U and there is a block sort of like you know you have kept your hand you are outside of the window and you can see that flow will go past as you are going in a car, obviously it is not a good idea to put your hand out from a moving car, but I just wanted to give this as an example.

So let us say, you have a flow past a cylinder and you want to predict the pressure, velocity x, velocity y, etc given the flow field. What you see here are contours of pressure and as well as velocity vectors, so you can see these velocity vectors here. So let us say you have done a lot of flows, so let us say you do flow 1 flow past a cylinder this gives you flow 1, then you do flow past a square all this using CFD and you keep on doing some simple shapes let us say triangle and we ask a certain question which is I give you a very complex shape let us say a car, what flow will it give? Or what velocity field will it give?

To repeat the question, we repeat we solve flows over elementary shapes going back to the previous simple example that I did in the previous video I had data for previous history of you know theta V, X and M and I gave you theta as the output. Now similarly I solved for various flows and I got the full flow field, now I want to know that if I have a new flow or a new body, what is going to be the flow?

So I leave you with this question, please think about what would be the appropriate input? What would be the appropriate output of this problem? What would be the architecture that we would use? What would be the loss function? So these are important design choices within deep learning. So I leave you with this question and I will come back with the solution as people have solved it in the next video, thank you.