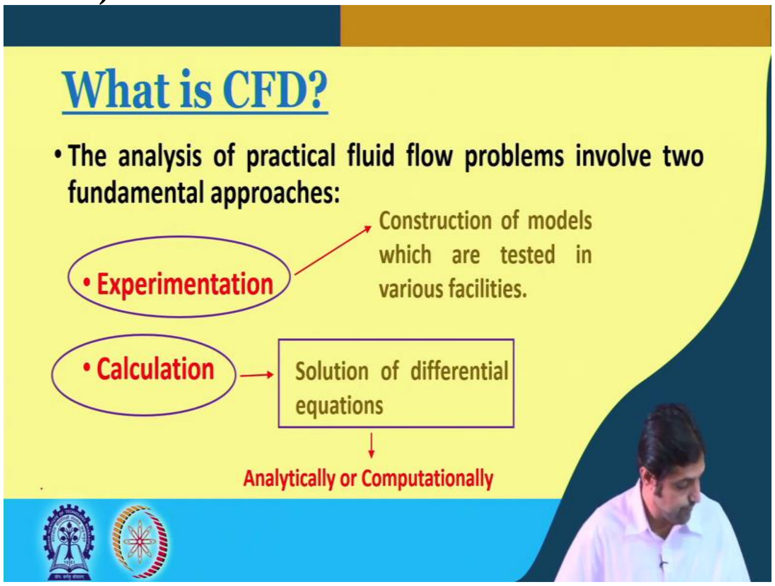
Hydraulic Engineering
Prof. Mohammad Saud Afzal
Department of Civil Engineering
Indian Institute of Technology-Kharagpur

Lecture # 54
Computational fluid dynamics

Welcome student to week 11 of course, hydraulic engineering. This week we are going to study topic called introduction to computational fluid dynamics. This is in continuation to the last week is module where we studies with viscous fluid flow and we derived the Navier-stokes equation. So, computational fluid dynamics is nothing more than solution of this Navier-stokes equation. The reverse stroke equation is quite complex.

So, there are different ways of solving those. So, this module is dedicated to that it says this is a undergraduate course we are not going to give you too many details, but at least touch upon all the basic concepts of computational fluid dynamics. So, let us get started with this.

(Refer Slide Time: 01:13)

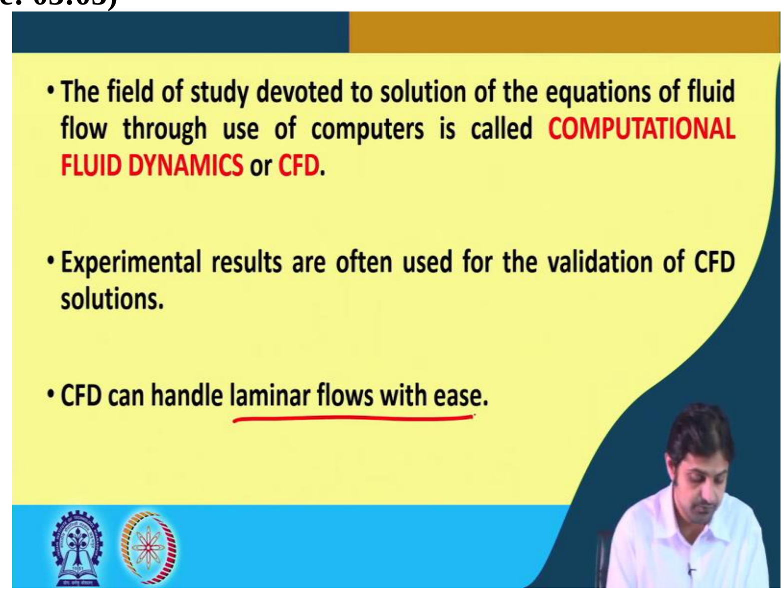


So, the first question that is very common and very obvious that you should have in mind is what is CFD so, the analysis of practical fluid flow problems involve 2 fundamental approaches. So, if you try to solve any practical fluid flow problems, we can use 2 approaches one is experimentation and the second is the calculation. So, experimentation requires the constructions of model which are tested in various facilities.

One topic we have already said and read in this course was dimensional analysis is very critical to this experimentation because, we when we do the experiments in the lab to be able to apply those in real life we need to study dimensional analysis and the as it is written here experimentation involves the construction of models, which are tested in various facilities, whereas calculation involves solution of differential equations.

Whatever the differential equation of the problem that we form that is the solution to that differential equation is called calculation. So, this calculation can be performed in 2 ways one is analytically or the second is computationally analytical is mathematical solution, for example, like you solve the integration is an integral equation by hand using the formulas of math computational is using the computers.

(Refer Slide Time: 03:05)



So, the field of study devoted to the solution of the equation of fluid through the use of computers is called computational fluid dynamics or CFD. So this is a very crude definition, but at your level we can say that the field of study devoted to solution of the equations of fluid flow through the use of computers is called computational fluid dynamics or CFD. So, the equations of fluid flow could be Isler equation Bernoulli’s equation.

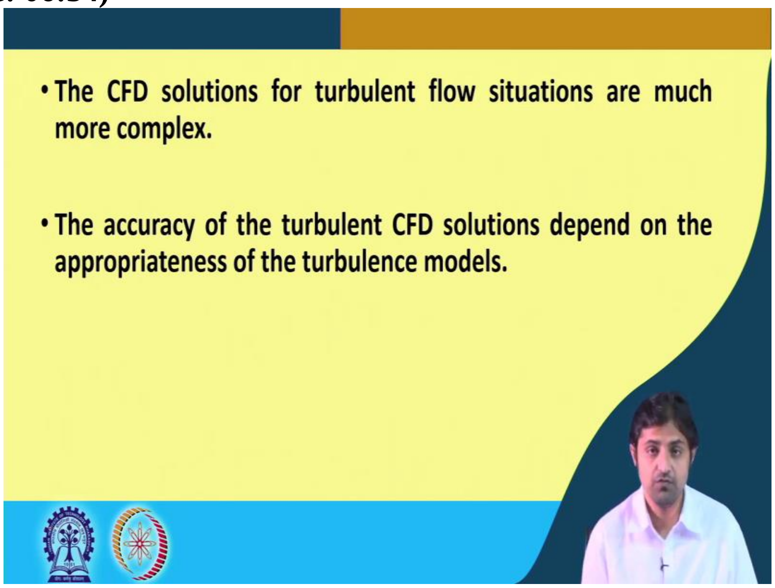
So, they are in a broad sense from the CFD but in reality, the solution of Navier Stokes it is called computational fluid dynamics or CFD, C stands for computational, F stands for fluid, D stands for dynamics. In this field of study called computational fluid dynamics, experimental results are used, and they are often used and what is the purpose of using those experimental results? They are used for validation of the CFD solution.

So if we get a computer based solution from the equations, we use computer to solve those equation and we get some answers results. How do we know if it is correct or not? So, what we generally do is we do the mathematical calculations for the same situation as one of the experiments had been done before either we do the experiment in the lab or somebody else might have done.

But beforehand we know the experimental results of a situation for the same situation we do math computer based solution and we may try to check if the results that we have got from the computer is the same or very close to the experimental results or not. So, this process is called validation means so, it will validate it will justify or this will say that, the CFD solution is ok or not.

So, if the board or the solution is successfully validated against experimental results or a real life field study, that process is called the validation of the CFD solution. So, computational fluid dynamics can handle laminar flows with very, I mean with ease, it is not a problem at all the problems become complicated when we have turbulent fluid flow.

(Refer Slide Time: 06:34)



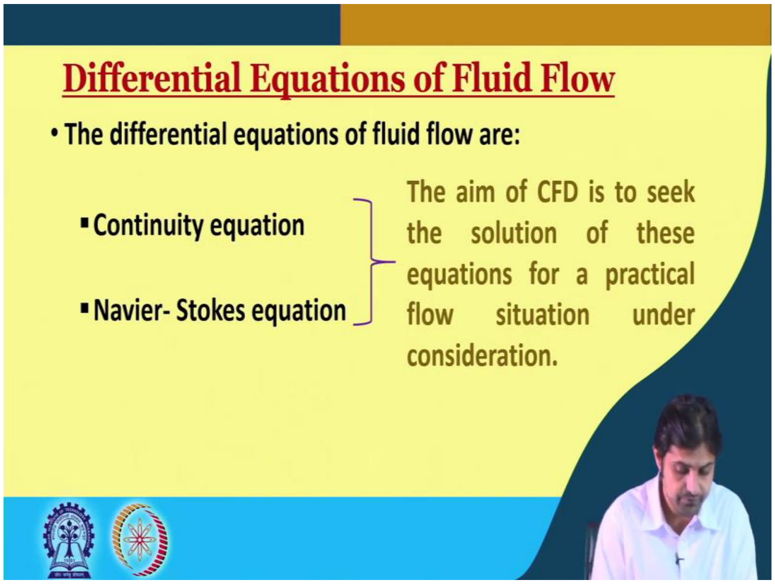
So, the CFD solutions for turbulent flow situations are much more complex all right. The accuracy of the turbulence CFD solutions depends on the appropriateness of the turbulence models. So, you will be late Study in this module that there are something called turbulence model just to name them one is you know k epsilon k omega there are different model. So, these are called turbulence model.

There is something called which cannot be actually said turbulence model, but, a different way of solving this technique is called largely simulation there is something called direct numerical simulation. So, the accuracy of these turbulence CFD calculation depends which module which model are we going to apply, we will see that direct numerical simulation gives the best results most accurate results largely simulations give the second best results.

And the last is Reynolds averaged navier stokes equation in Reynolds averaged navier stokes equation there are different turbulence modules. But everything comes with a cost. For example, you must just know that direct numerical simulation although it has the best accuracy, but the time of computation that is required is extremely high. So, largely simulation are the second best the act in terms of accuracy, but cost wise it is also very high Of course, less than direct numerical simulation.

Whereas, Reynolds average navier stokes equation has less than accuracy than the above 2, but the time of computation is far less than what these 2 type of models in CFD takes, but we will come to those later as well. So, this is what it means the accuracy of the turbulence CFD solution depend upon the appropriateness of the turbulence model, which turbulence model is appropriate.

(Refer Slide Time: 08:49)



Now, going to differential equations of the fluid flow So, the differential equations of fluid flow are we have been going through this for a long time now, one is continuity equation that we have seen for example, incompressible flow

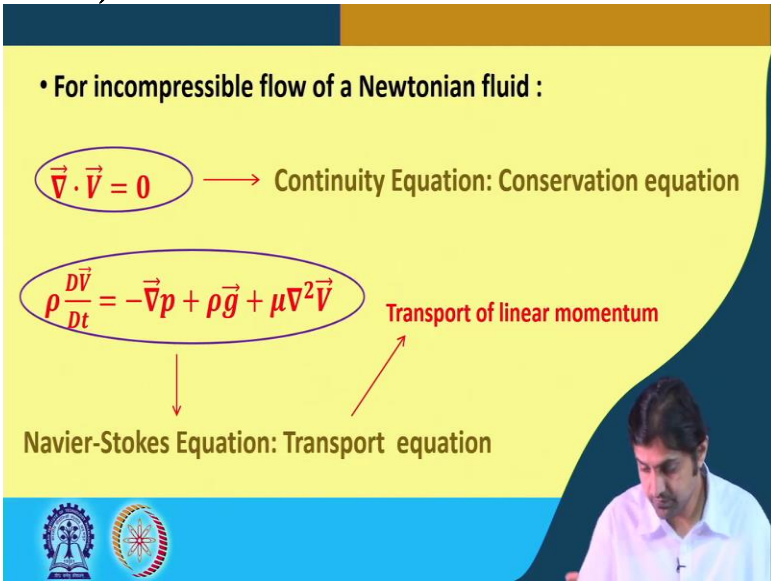
$$\vec{\nabla} \cdot \vec{V} = 0$$

, which you have seen actually in the last week is model with lectures on viscous fluid flow and the second is the Navier stokes equation. So, these are the 2 equations differential equations of fluid flow.

And the solution of these equations through computers is called CFD computational fluid dynamics. So, the aim of CFD is to seek the solution of these equations for practical flow situations under consideration. You remember when we started this lecture, we defined CFD the use of computers to solve the fluid flow equation. So, these are the fluid flow equation that we are talking about continuity equation.

And the Navier stokes equation and the aim of CFD is to seek the solution of these equations and it should be for a practical flow situation we should not assume say a velocity of 1 lakh meters per second or something like that, no, I mean the this the simulation should be very much practical in nature like a velocity of 1 meters per second or a wave traveling with a wave height of 1 meter if we assume a wave height of like hundred meters, it might not be very real for example.

(Refer Slide Time: 10:43)

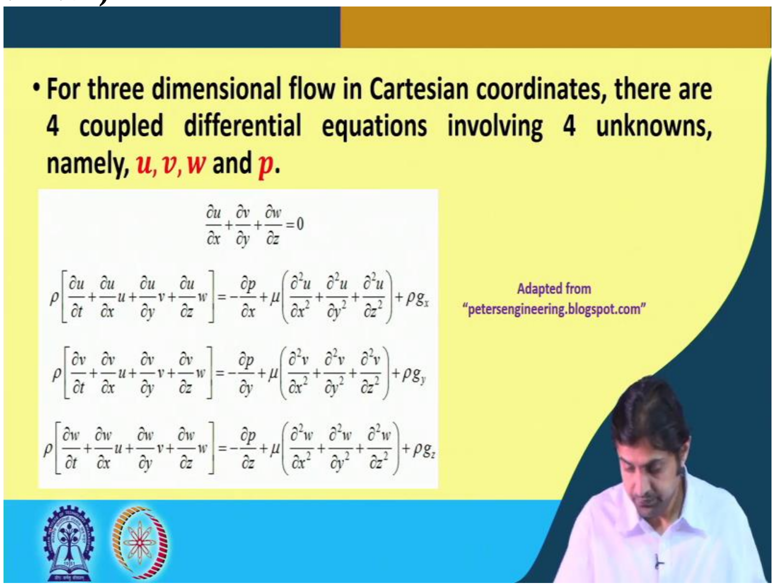


So, for an incompressible flow of Newtonian fluid this is the continuity equation

$$\vec{\nabla} \cdot \vec{V} = 0$$

and this is continuity equation or the conservation equation. The second equation if you can remember what is this yes this is navier stokes equation which is actually a transport equation and this navier stokes equation is for incompressible flow we want this one. So, as I said navier stokes equation can be classified as transport equation and continuity equation can be classified as conservation equation. So, this transport equation transports what it is a transport of linear momentum.

(Refer Slide Time: 12:04)



So, for 3 dimensional flow in Cartesian coordinates, there are 4 coupled differential equations involving 4 unknowns. So if there is a flow which is occurring in 3 dimension, there are 4 different equations, which involves 4 unknowns. So one of the unknown will be the velocity new direction, the other is going to be the velocity in redirection and the third one is going to be velocity in w direction.

All 3 directions are velocities, and the fourth one is going to be the pressure. These are the 4 unknown as it is already written here u, v, w and p. And if there are 4 equations, I mean, there are 4 variable value variables u, v, w, and p. There are 4 coupled differential equations. So, these are the 4 equation. So, this is nothing but $\vec{\nabla} \cdot \vec{V} = 0$ as it is returned $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$.

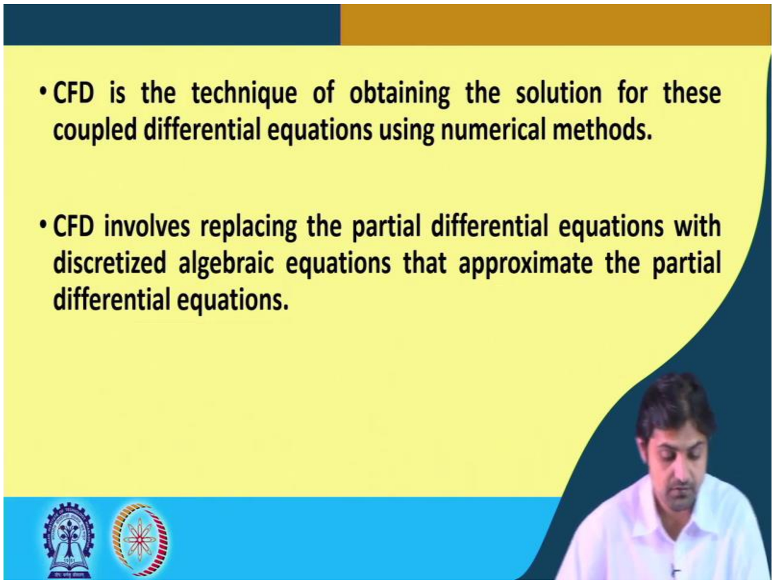
So, now you would understand why I taught the viscous fluid flow lecture before and we derived it everything, because, this particular module CFD comprises of many visions and it is it will help you a lot if you can recognize these equations. So, now, you can easily remember that this was equation of continuity, this one

$$\rho\left[\frac{\partial u}{\partial t} + \frac{\partial u}{\partial x}u + \frac{\partial u}{\partial y}v + \frac{\partial u}{\partial z}w\right] = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) + \rho g_x$$

. This is Momentum in x direction.

Similarly, and this actually we have derived all these equations right we wrote it in a general form in terms of i and j but if you write it in i j and k separately there is going to be 3 equations navier stokes equations for incompressible flow. So, this is momentum in the y direction and this is momentum in z direction.

(Refer Slide Time: 14:55)

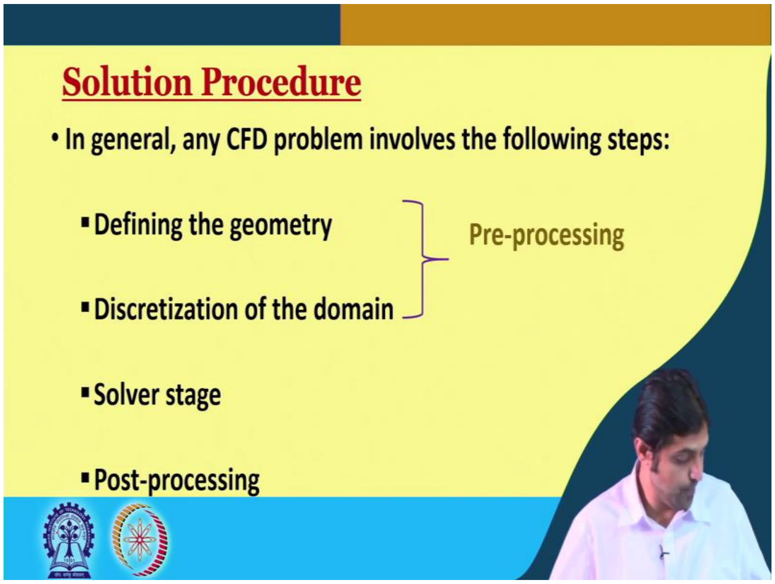


CFD is the technique of obtaining the solution of these coupled differential equations using numerical methods. So, we are refining the C definition of CFD as we are going on initially we describe we said that CFD is the computer solution of the flow equations and flow equations can be many right even Isler equation can be the solution I mean the flow equations, then we went and said CFD is the technique of obtaining solutions for momentum and continuity equation.

And now, we say that these equations are copper differential equations, and we use solve it using which methods numerical methods. I am hopeful that in you are B.tech course, you have studied a course called numerical methods in engineering, where the different techniques are taught. We will go to the basics of that, some of them in this module as well. CFD involve the phrase replacing the partial differential equation that is PDE with discretized algebraic equation.

That approximates the partial differential equation for example, I mean will So, for example, if they there is dp by dx, so, we have to replace it with for example, delta p by delta x or P2 - P1 divided by some things like that this is very basic to right but I hope this will make it clear to you what is discretized algebraic equation. So, this is like an algebraic equation. So, CFD involves replacing the partial differential equations with discretize algebraic equations that approximate the partial differential equation so, that is why there was a proximate sign when I wrote those.

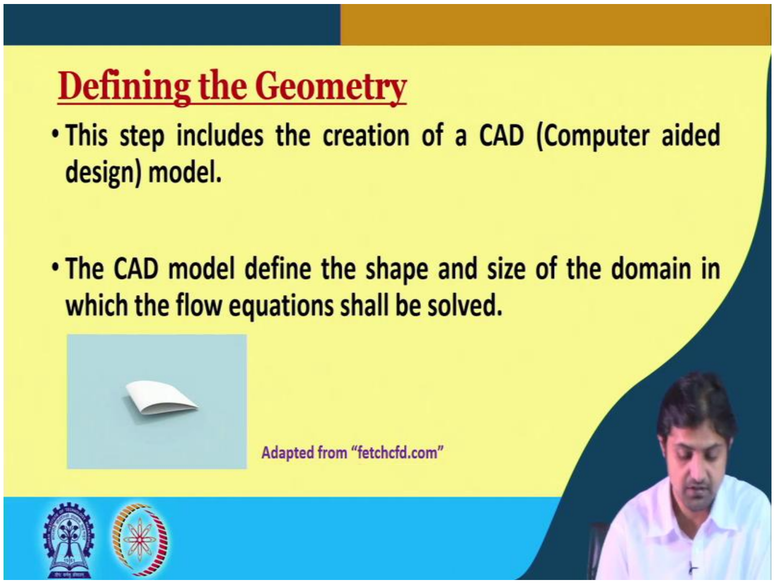
(Refer Slide Time: 17:07)



Now, what is the solution procedure? The solution procedure in general any CFD problems involves the following steps. First is we have to define the geometry of the flow, we have to discretize the domain we will come to it what defining the geometries what discretization of the domain is and then there is a solver stage. And in the end after the solution is solved, there is post processing, is after the results are obtained, we have to show it graphically or we have to find some values we have to interpret those results that we got in most of the cases plotting the results is termed as post processing.

So, defining the geometry and the discretization of the domain, these 2 things are called preprocessing. So, something that you have to do before the calculation can begin in the computer, all right. And the post processing is something that you have to do after the calculations have something to be done with the results after calculations have finished whereas preprocessing steps to be completed before the calculation can begin on computer. So, this is the general definition of preprocessing and post processing.

(Refer Slide Time: 19:29)

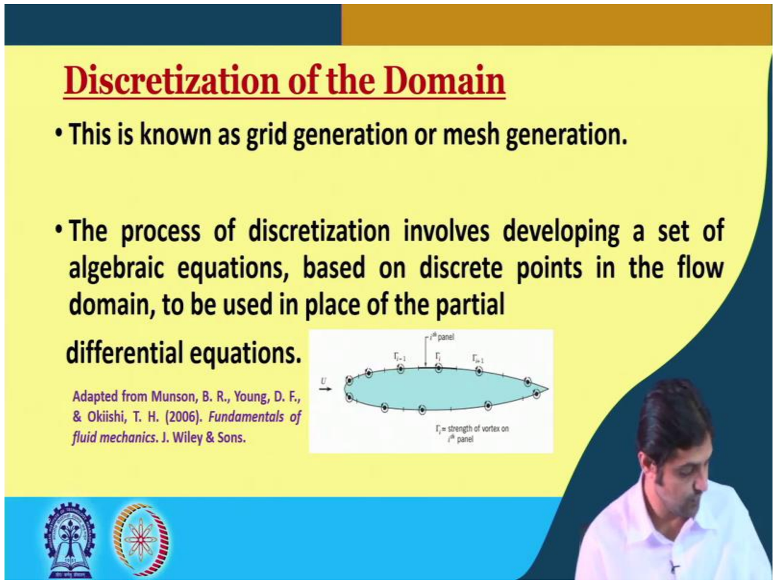


So now as we said that the first step is defining the geometry. So, this step includes the creation of a CAD model what is CAD computer aided design. So, you use some software is are there are even tools available within the computational fluid dynamics models where you can define the geometry or derive for example, suppose for example, there is a tank right and there is a pier. And you have to and this is open no I am just drawing it in 2d assume it is 3d and flow is coming through this right.

So, you will have to define this tank and this so, you can do it using the CAD models and this is defining the geometry. So, what does this CAD model do, the CAD model define the shape and size of the domain in which the flow equations shall be solved so, what do they do the CAD model define the shape say for example, as we said it was rectangular in shape right and the size of the domain.

So, what the length of the for example and what would be the water depth for example, here water depth or mean your depth of the tank actually not the water depth I love this one and also the width in 3d shape and the size of the domain. So, for this is a CAD model for you see this resembles the wing of an aeroplane for example. And this has been taken from fetchcfd.com this image that is why we have given a reference so as alright.

(Refer Slide Time: 22:07)



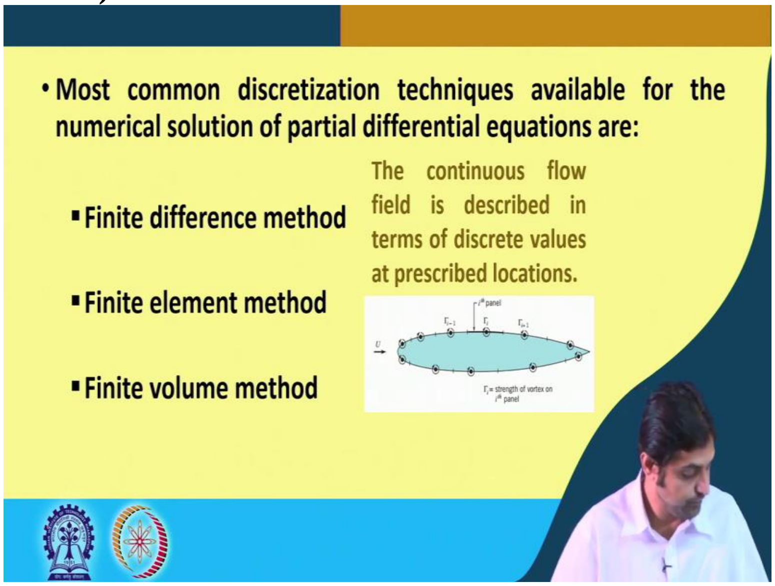
So, the second step was discretization of the domain. So, this process is known as grid generation or mesh generation. So, the process this particular process of discretization involves developing a set of algebraic equations based on discrete points in the flow domain to be used in place of partial differential equation we told that the way that it is to be done instead of partial differential equation we try to transform it into algebraic equations.

So, for that for example, if this is a you know domain we need to know so, suppose we have to divide it right that is how we are going to ride delta x and we have to divide it in so let us say We have divided into so many different parts in x and y direction. So, pardon my brain. So, the process of distribution was developing a set of algebraic equation based on discrete points in the flow domain.

So, you know, we will so, this is what meshing is, so, we divide our domain into so many small parts and these are helpful, because then we can actually use of these points the if you remember if we if you remember from your limits class suppose del p del x can be approximated as P2 - P1 divided by x 2 - x 1. So, this we always used to right limit delta x goes to 0 right. So, this means that the grid should be as small as possible for this process to be done the which process that converting the partial differential equation in into the algebraic form.

So for example, if there is a flow U coming from this side we have defined 1.2 point 3456787. So it i is the strength of vortex and the ix panel i x d i, d i d x + 1. So, this is the way that we do and we have taken this figure from m2 and m1 is official. So defining these points or forming those points using the help of computer is called the mesh generation.

(Refer Slide Time: 24:44)

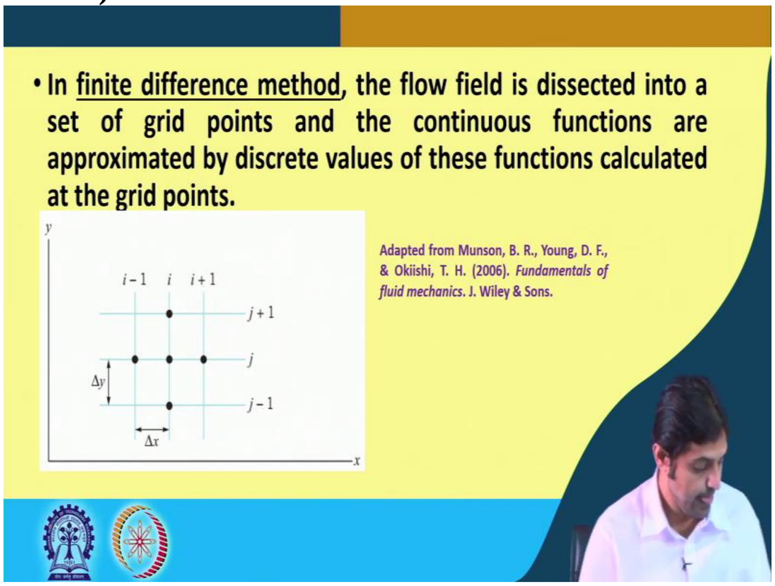


So, the most common discretization techniques available for the numerical solution of partial differential equations are the finite difference method. The second is finite element method and another one is finite volume method. So, we will go into small details of this, so that you have a broader idea, you will not become expert overnight and all these techniques, but will give you a very, good understanding and overall a holistic picture of what computational fluid dynamics is. So, we will touch briefly all of these topics.

So, as I said that these are these 3 are some of the most commonly used discretization techniques available for numerical solution. So, this is the second step in this CFD solution first was the which was the first one if you go back and you see, the first one was defining the geometry. The second one was discretization of the domain and as I said, discretization of domain can be done using 3 different methods.

First finite difference method, second is finite element method and third is finite volume method. So, if you see to this figure, the continuous flow field is described in terms of discrete value that describe location.

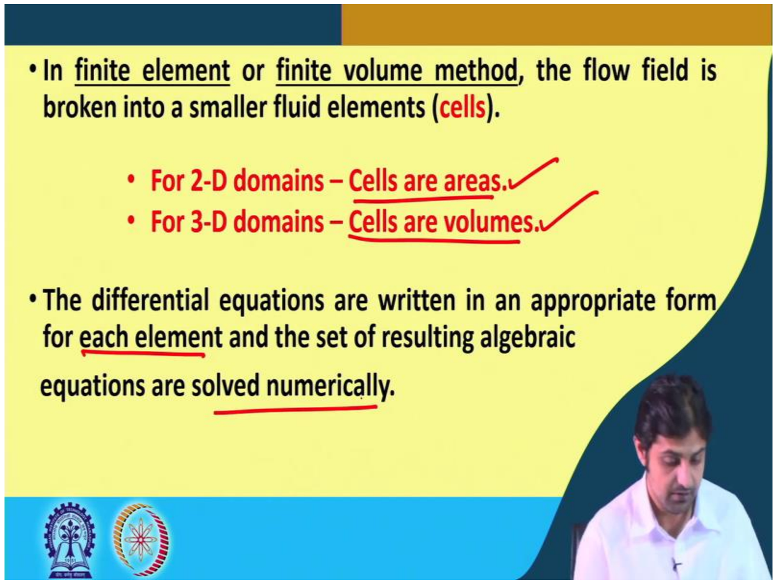
(Refer Slide Time: 26:22)



In finite difference method, the flow field is dissected into a set of grid points. And the continuous functions are approximated by discrete values of these functions calculated at grid point. So, what happens in finite difference the flow field is dissected into a set of grid points as I showed you, right so, this is 1 grid or these are grid points in x direction this is grid in y direction and the continuous functions are approximated by discrete values of these functions.

So, what we do we calculate the value suppose the velocity and pressure we will calculate at this point we will calculate at this point using the our conversion of partial differential equation to the algebraic equation. This is the thing that I was trying to explain. So, if this is the delta x has been the mesh size or grid delta y. So, this is this method is called in finite difference method. If this is I this is i + 1 this is - 1, this is you know suppose this point is I, j. So, this will be this point is going to be i, j + 1, this point is going to be i + 1, j + 1 now this is how it is defined. So, this is I, j - 1, this is i + 1, j - 1.

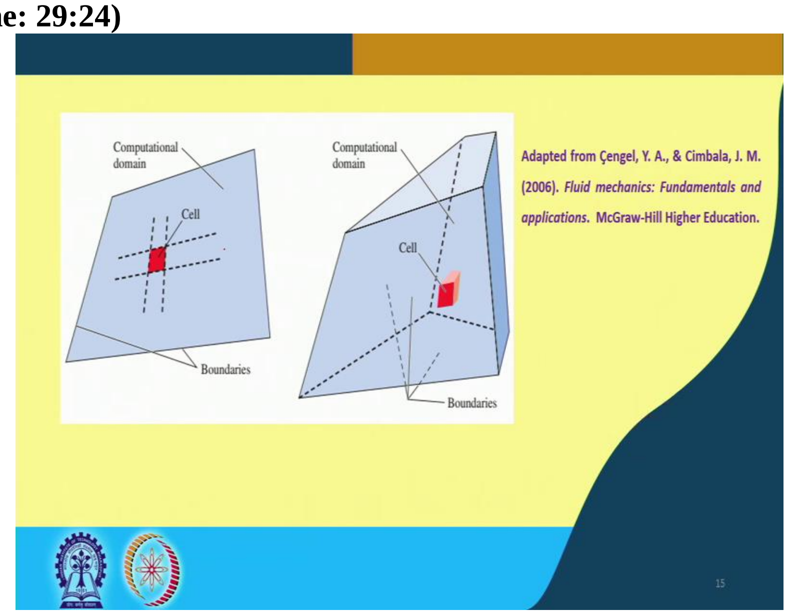
(Refer Slide Time: 28:28)



So, in finite element or finite volume method the flow field is broken into a smaller fluid element called cells. Alright. So, for 2D domain cells are areas all right, and for 3d domains, these cells are volumes. So, the differential equations are written in appropriate form for each element and this set of resulting algebraically equations are solved numerically. So the differential equations are written for each of these elements.

And the set of resulting algebraic equations are solved numerically, so we write this equation for each of these elements if there are thousand element we write thousand equations, and we solve it numerically.

(Refer Slide Time: 29:24)



So this is an example where you see it is a 2d domain and therefore this is a cell. In the computational domain, it is 2d. This is 3 dimensional come become computational domain. And you see, these are like volumes or you can also think in a way that this 2D, I mean, for a same object, we have shown a 2d representation, we can say and we have shown a 3d representation. It is up to you how you want to take it.

But the main message of that needs to be conveyed here is that for 2d domains, the cells are areas and for 3d domains, the cells are volume. So, I think this is a nice point to stop this lecture, and we will resume in our next lecture from this point. So, thank you for listening. I will see you in the next class.