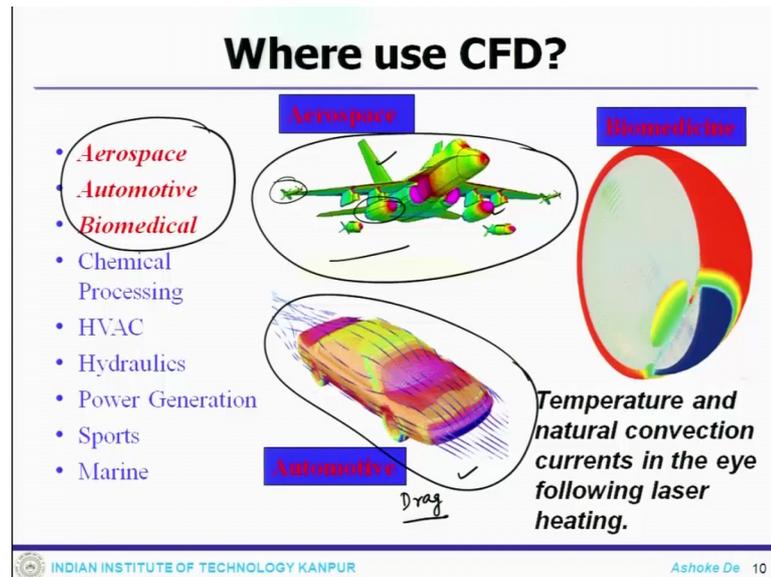**Introduction to Finite Volume Methods-I**
**Prof. Ashoke De**
**Department of Aerospace Engineering**
**Indian Institute of Technology, Kanpur**

**Lecture – 02**

(Refer Slide Time: 00:15)



So, welcome to the lecture of this Finite Volume Method. Now where you have used? These are the certain examples that I would like to get your attention, but this is not all. This is just some examples, but the application area as I said it is spanned over microchannel to microfluidics to nano fluidics to the space that is the span. So, if you look at this first this particular slide, we will have a look at the example from aerospace. As I have been repeatedly saying that this is the solution that we can do, the flow around the aircraft, this is what aerospace industry wants. The, this is the complete aircraft that we are talking.

So, flow around that taking consideration of all these nitty bitty details like wing associated with this small structures these engines. So, these are all individually equally complicated system, but we can get a simulation done not only for these individual components. We can get a complete solution done around this. Similarly, we can get a solution done around a space shuttle. So, this is the application of aerospace industry looking from. And today using the CFD technique, one can even get a solution done

from a engine. Engine is another complicated system integrated system for the any vehicle; from end to end solution is also possible.

You look at the automobile engines. This is only showing a picture of the flow around a automobiles, but that is not all. So, there are other component like the ventilations, air conditioning, these are all indoor problem of the automobile system engine, because engine is another key component to the automobile sector. Everything starting from the engine starting from the external design so, this will get you the fluid flow systems around the vehicle, and that you get you the estimate of the drag. So, you can design your vehicle more efficiently. So, this is what over the decade or couple of decades that has done.

I mean if you look at the now the design of automobile or the vehicles or the car, now the design has gone more aerodynamically design; that means, the drag has reduced so, that means, more efficient. So, once the drag has reduced the fuel consumption will be reduced. So, this has more economical the price will come down. So, this has a kind of closed loop implications how you design the system, ok.

Now, you look at the biomedical applications. This is an example of a distribution of temperature in some laser heating of a eye. Now, in a biomedical system also the CFD can be applied and CFD application is in normal. Not only the eye you can even look at the flow through your arteries, you can look at the flow through your veins, you can look at the flow in your hard walls, you can look at drug delivery. So, all these things can be applied in the biomedical system and this is only possible due to CFD.

(Refer Slide Time: 03:25)



Now you look at the chemical processing system. This is a reactor vessels so, reactor vessels you just prediction of the flow separations. So, this is what the chemical process industry would be interested. They would be also interested look at the mixing process, because one of the key component in the chemical industry is the mixing. Because all these drags and all these things are done that kind of process and mixing is another chemical phenomena which chemical industry would be looking at it.

Now, HVAC system this is the ventilation system; this inside a room when the person is sitting in front of a computer. So now, when you talk about the architecture, we talk about the application of CFD, we talk about that CFD can be used because of the advancement of the computer. Now we have high end computers; like, we have a million of cores I mean processors essentially we talk about the high end computer means processors. And those processors are housed in a data center.

In every industry academics, all where they have a data center which can house this kind of computer. Now, once you how those kind of powerful computers or cluster or workstation, they need ventilations and that ventilation mean essentially you look at the air conditioning system. So, that is again the application of CFD. One hand you talking about the architecture of the computer, other hand you talk about the cooling process of that system, now cooling also needs to be done efficiently so, again you use CFD to get the cooling done. So, that is one application now when you talk about the architecture

inside the computer even each and every chip the cooling is required. So, that cooling is another interesting problem, and which can be solved through the CFD.

You look at the problem which is applicable to civil engineers, the hydraulic problems. So, there again you can solve through CFD. So, the picture is not restricted to this list, but it has wide range of applications.

(Refer Slide Time: 05:26)



Again if you look at the sports, the person who is swimming inside the pool so, you can actually look at the dynamics of that thing. The reason is that when a swimmer actually swims he wants to move faster. And the way he can move faster is that how he can moves his body, or how he can she can propel his body faster. So, he has to or she has to reduce the drag move the flow around such a way that he or she can more faster. Again CFD would be very handy to get a solution done.

And even today, if you look at literature if you look at the swimming of the micro organs, swimming of the fees and this can be represented through CFD. Marine applications, again there is a huge large scale problems, and one can get a solution through CFD. Power generation; that means, power plants this cooling towers. This has a multidimensional problem. So, you can look at how the flow around this cooling towers what is the interaction between the fluid and the structures. So, this is also certain lead to the certain fluid structure interaction problem. So, if you look at it now things become multi-disciplinary.

When you talk about fluid structure interaction problem means, which is a flow around this towers flow around the swimmers flow around the flapping wings, all these are associated with the fluid structure interaction problem; that means, you have to solve a fluid problem, you have to solve a structural problem their deformation their inter coupling.

So, that means, it could be done one way coupling it could be done strongly. Coupling the effective solution or the desired solution would be getting a solution done through the 2 way coupling. So, that will lead to the fluid structure interaction problem. And all these are the under the resume of the computational fluid dynamics. So, these are the area of applications one can expect where CFD can be used. Now, when you talk about that what is the CFD need to the physics of fluid. So, what physics of fluid?

(Refer Slide Time: 07:44)



Fluid is essentially either it could be liquid or it could be gas, when you say that these are the only form of liquid. Now there are certain properties of the fluid one is one important property is the density. And when the density remains constant or density does not vary with time then or space also we call it a incompressible system. Or the density becomes variable it is a compressible system or variable density system, variable density flow.

So, both the cases density is varying. There is another property which is called the viscosity. So, viscosity is essentially the resistance of the fluid. As we know the fluids actually cannot resist the shear stress. So, that resistance comes through the viscosity.

Now if you look at the viscosity density these are properties which are primarily dependent of temperature. And there are other transport properties. See, this is one of the transport property. We call it a transport property. There are other transport properties. Like diffusivity, thermal conductivity. So, these are all transport properties so, when you deal with a multi species system, then you come across with the diffusivity.

Individual species like when you talk about air compressor with nitrogen, oxygen, carbon dioxide multiple species. So, diffusivity of individual species are different. And when you talk about those diffusivity, there also certain transport properties. Thermal conductivity, these are also transport properties.

(Refer Slide Time: 09:40)
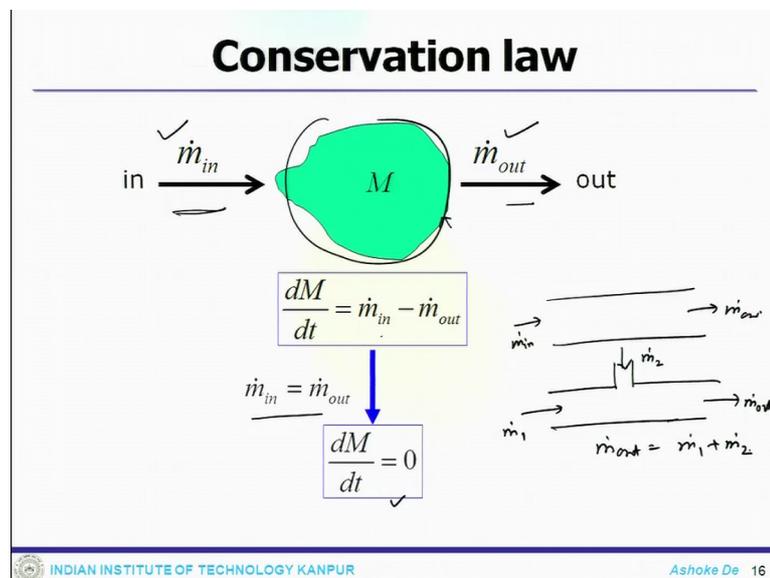
They are somehow or other way related to the system of temperature. Now what are the conservation laws that are required to represent the fluid system? One of the very, very basic and fundamental law that requires to govern a fluid system is the mass conservation.

So, the system is that if you have a system like this, the mass coming in and mass going out. The conservation says the change of the mass in this particular isolated system is the difference between the mass flux. And this has to be conserved. Because if your system is in place, then the mass coming in and the mass going out they need to be balanced. So, the dM dt needs to be 0. So, then the mass balancing is done.

If that does not happen, it is just like a you have flow through a channel. There is a mass coming in; there is a mass going out. If the mass balancing is not done, then there is no conservation of the mass law; that means, there will be somewhere and it is also applicable to any multiple inlets, you can have multiple inlets. This would be a m dot 2, this could be m dot 1; that means, the 2 inlets, but the outlet is.

So, the mass conservation says that this has to be satisfied. If this does not satisfy; that means, there would be some accumulation, that cannot happen in a realistic problem; that means, there is a accumulation of the mass. So, it violates the basic principle. So, the one of the important thing is the mass conservation has to taken place. And the thing is that mass momentum energy these are the 3 component that will be talking about.

(Refer Slide Time: 11:27)



Now, when you talk about the mass conservation, that leads to the continuity equations. And continuity equation in the form of this, del rho by del t D rho by Dt, this is called the material derivative, or the property, and this is the (Refer Time: 11:46) that these applies to incompressible flow. When it comes down to incompressible flow, the variation of the density is not there. So, rho is constant and you get back the incompressible system. So, mass conservation lead to the equation of the continuity so, this is one of the equation that we require, ok.

(Refer Slide Time: 12:10)



## Navier-Stokes Equation II

➤ Momentum Conservation→Momentum Equation

$$\rho \frac{\partial U_j}{\partial t} + \rho U_i \frac{\partial U_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_i} + \rho g_j$$

$$\underbrace{\rho \frac{\partial U_j}{\partial t}}_{I} + \underbrace{\rho U_i \frac{\partial U_j}{\partial x_i}}_{II} = \underbrace{-\frac{\partial P}{\partial x_j}}_{III} \underbrace{- \frac{\partial \tau_{ij}}{\partial x_i}}_{IV} + \underbrace{\rho g_j}_{V}$$

I : Local change with time
II : Momentum convection
III: Surface force
IV: Molecular-dependent momentum exchange(diffusion)
V: Mass force

$$\tau_{ij} = -\mu \left( \frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) + \frac{2}{3} \delta_{ij} \mu \frac{\partial U_k}{\partial x_k}$$

INDIAN INSTITUTE OF TECHNOLOGY KANPUR                    Ashoke De   18

Now, momentum conservation so, this is talking about the momentum conservation. So, momentum conservation get back to the momentum equation. So, this is the equation it looks like. So, at this moment you just talk about the equation, then in the follow up lectures we will how these equations are actually derived. There are multiple terms, the first term which is associated with that the local change or with the velocity with the time.

The second term is the convection of the velocity third term is the surface force or the pressure gradient term. 4th term is the diffusion term, and this is the source term due to gravity. So, the source term could be for something else also. So, one individual momentum conservation equation actually contains so many terms.

(Refer Slide Time: 13:01)



Now, for incompressible fluid if you look at it the shared system is represented like that. And then the conservation of continuity this is coming from the mass conservation equation. So, this is coming from mass conservation equation, and that leads to this particular system. And the incompressible momentum equation boils down to this simple form.

So, all this equation will again derived and we can see for a particular control volume how we get back this equation. Because these are the actually key component of the system and you need to understand these things. Energy conservation equation here it is shown in form of temperature.

(Refer Slide Time: 13:50)



But while we will be talking about the energy conservation equations, we will see we can have a different form of energy conservation equation. Again if you look at the multiple terms sitting in the left hand side and the right hand side these are the term, which are associated at the left hand side. And these are the term, which are associated at the right hand side. Left hand side you have a unsteady term you have a convection term, then you have a pressure based work done term, this is the diffusion term and this is the energy term which is associated with the system.

(Refer Slide Time: 14:33)

Now, we talk about the mathematical system, which get you the discretized system. Now this is a physical problem that we are dealing with you think about it. Now you want to solve the physical problem on that. So, we can have a analytical equations or the differential equations. Through the numerical techniques or the discretization technique this is where your numerical technique would come in picture. Through the numerical technique you get a set of discretized equations. And these discretized equations will essentially lead to a set of linear system. And this linear system is through the programming you feed the set of instruction to the computer.

So, you have a physical problem in place. You bring down that physical problem to the linear system, linear system through the programming you feed it to the computer. The computer will get you the solution of this particular problem. Now, when you talk about these numerical techniques there are different different techniques. One of the important technique is or the very fundamental technique is the finite difference method. So, in a what one can say that it is much more straight forward to apply and simple. But the applicability is restricted to the structure grid. The structure grid, how it looks, what they are what kind of complications or simplification they have, all these things we will talk once we talk about the different grids.

 Second category of the discretization method could be finite element method. It can be applied to any geometry. That is one of the biggest advantage over FDM. This is advantage now final is that finite volume method. This is also can be applied to any geometries. So, if you compare to finite difference over finite element or finite volume. Applicability of finite difference volume and finite element are more wide because they can be applied to any geometry. And that is why in a fluid flow system finite volume methods are widely accepted and used starting from your academic work to the industrial applications. Finite element also being used, but it is not that popular.

(Refer Slide Time: 17:05)



## Discretization

| Name of the Method | Process | Advantage | Disadvantage |
|---|---|---|---|
| Finite element Method (FEM) | It finds solutions at discrete spatial regions (called elements) by assuming that the governing differential equations apply to the continuum within each element. | ▪ Successful in solid mechanics applications.<br><br>▪ Their introduction and ready acceptance in fluid mechanics were due to relative ease by which flow problems with complicated boundary shapes could be modeled, especially when compared with FDMs. | ▪ More complicated matrix operations are required to solve the resulting system of equations<br><br>▪ Meaningful variational formulations are difficult to obtain for high Reynolds number flows<br><br>▪ Variational principle-based FEM is limited to solutions of creeping flow and heat conduction problems |
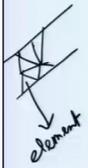
Now, if you put them together this techniques. So, as I said one is the finite difference method. So, if you look at the process, the methods essentially assumes or include certain assumption of the variation of the unknown, which would be computed over different grid points. So, if this is the problem I am trying to look at through finite difference method, it will get me the solution at all these points. So, essentially finite difference method gives me back a solution which is a nodal solution.

So, advantage it is a very straight forward. It is quite simple to use and whoever is a newcomer to the field he should be able to get a solution or idea about the numerical techniques quite easily. But disadvantage one of the major drawback of this is that it is not at all suitable to the solve problems with physical complexity; that means, if I have a arbitrarily shaped problems. Then I would not be able to apply it just like a you think about flow around your complete aircraft structure or flow around your automobile vehicle or flow around your chemical process industry. You cannot actually apply finite difference method, because they cannot be applied.

. Secondly, it is solutions are also very time dependent, strongly time dependent and at high Reynolds number flows there are certain limitation. While you go back to finite element method, it is also find solution in the spatial region called the elements. So, if I have a plate like that I can get solution like this. These are the individual component of

this these are called the element; you get a solution of this kind of a finite element applying your continuum mechanics.

Advantage it is quite successful in the application of the solid mechanics, but the application in the fluid mechanics has been evolving and the application is bit restricted, but it is not that popular compared to finite difference or finite volume method. Disadvantage it requires complicated matrix operation; that means, the solution of the linear system is going to be complicated or time consuming, then meaningful variational formulations are difficult to obtain for high Reynolds number flows. Then variational principle based finite element is limited to solution of very creeping flow or heat conduction problems. So, these are certain restriction which actually restricts this finite element to be popular for fluid flow heat transfer related problem.
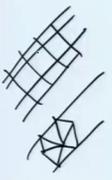
(Refer Slide Time: 20:17)



## Discretization

| Name of the Method | Process | Advantage | Disadvantage |
|---|---|---|---|
| Spectral Method | The approximation is based on expansions of independent variables into finite series of smooth functions. | It can be easily combined with standard FDMs. | ▪ Their relative complexity in comparison with standard FDMs<br>▪ Implementation of complex boundary conditions appears to be a frequent source of considerable difficulty |

INDIAN INSTITUTE OF TECHNOLOGY KANPUR                    Ashoke De   25

There is another kind of technique which are also known is the spectral method. This is more based on the smooth function or the series of finite series of smooth functions which are used to represent the variables. And it can be easily combined with standard FDMs, but major disadvantage is that it also cannot be applied to complicated problems. And when you talk about the fluid flow problem, you are kind of exposed to a different different boundary conditions; that means, these are difficult boundary conditions and implementation of those kind of boundary condition in spectral method is always with difficult task.

So, that is why this is also not very popular. In widely popular and accepted method is the finite volume method.

(Refer Slide Time: 21:17)



**Discretization**

| Name of the Method | Process | Advantage | Disadvantage |
|---|---|---|---|
| Finite Volume Method (FVM) | ▪ Domain is divided into a number of non-overlapping control volumes<br>▪ The differential equation is integrated over each control volume<br>▪ Piecewise profiles expressing the variation of the unknown between the grid points are used to evaluate the required integrals | Physical soundness | Not as straightforward as FDM |

So, what it does is that you divide the system in number of control volumes just like if I look at this channel, then these are the control volume. It could be like that; it could be like that too. So, these are the finite control volume. So, the governing equations or the set of partial differential equations they are solved over this finite volume, ok.

Advantage is that, physically very sound and it can be applied to any sort of geometries, but slight disadvantage is that it is not that straight forward. But this is one of the method which is very well accepted in fluid mechanics community for the CFD applications. So, and what we will be discussing about this particular method. So, when I said all this different technique that mean one can use any of these techniques to get a CFD solution done. So, CFD is not only talk about finite volume method, but these particular lecture series we will only concentrate on the finite volume method.

(Refer Slide Time: 22:29)



Now, if you look at the again coming back to the just give you what finite volume does in a quick fashion. The general solution of the Navier-Stokes equation or the general form of the Navier-Stokes equation, this is the phi could be any quantity one this corresponds to continuity equation, this corresponds to momentum equation, this corresponds to temperature equation. So, any variable in terms of phi, you can write the transient, you can write the complete flux term, this is local change with time, this is the flux and this is the source term.

And if you integrate over a control volume, this is how you are going to get it. So, and the integral form of the Navier-Stokes equation will look like this is a local change inside the control volume. This is the flux over the control volume surface; this is the source in the control volume. Now, once you bring down to this particular system; that means, if you look at it this is my governing equation, and this is where the techniques brings me down in this particular form. Now, when you do the conservation, this is at the 2 particular finite volume the simple a and b 2 volume.

(Refer Slide Time: 23:47)



So, these are the direction of the fluxes which are shown, this is incoming fluxes and to get a complete set of domain or the grid, these are all individual volumes. So, you can think about the individual volumes for all these individual volumes, this equation is valid. So, essentially the conservation laws which is integrated over individual volume so, the dV actually represents a individual control volume of the control volume. Now once you integrate over that this is the complete set of equation that you end up getting.

(Refer Slide Time: 24:35)

Now, if you look at the surface so, essentially this is my one finite volume, the center of the finite volume if you say P, then the general convention for the finite volume which will be consistently using throughout this lecture, the ahead of it always a east side of it upper is not typical directional concept. There is a behind of the, this particular volume is west lower is south. So, one particular control volume is connected with some neighbouring control volume; one is east, north, west, south. And the phases which are kind of connected to each other the between the P and E this is E surface. Between P and N, it is a north surface, between P and west it is west surface, between P and south is a S surface.

And the component so, if you have approximation of the volume integral like the mass over rho dV. So, you can approximate like a density of this particular volume. So, rho P is the density of that particular volume and the volume V. So, the V is the volume of this particular element the element P. And mu would be the rho i u i dV so rho P u P V. So, basically that is the momentum approximation.

If you use the approximation of the surface integral, then this is the delta P dV. So, if I convert that integral to the surface integral. It is P dS and then if I take the surface integral along all these surface, if you look at the element P, this P is surrounded with 4 surface, because it is a 2 dimensional element. So, the surface east surface west surface north and south surface, that is why the integral is complete the summation over all the surface. And k goes from north south east west. So, essentially the volume integral over this particular element actually brings down to the surface integral or the summation of the surface.

So now once you do the interpolation. So, it depends on the flux, if my velocity moved from the these to this direction; that means, P element to E element, then the this particular interpolation scheme. Do not worry about that too much because we will be talking about all this in detail. This will be taking the sale of E. If the velocity in the other way around, then it will be taking the information of the this element; that means, this particular interpolation scheme for velocity actually takes into account the local direction of the velocity. Either the flow can go from P element to E element, then the velocity of this surface would be U P, if the flow goes from E element to P element the velocity of this surface would be U E.

Now, if I use a different kind of scheme, that the central scheme then I will take the some interpolation between P element and u element like this. So, if you look at the systems, like in the previous system, we have this equation and this equation brings down to this kind of new mathematical expressions.

(Refer Slide Time: 28:14)



So, finally, around one particular control volume; that means, the for that element P. Element P if I write the equation, this is how my equation is going to be look like; that means, these are the different coefficients.

So, how you get this coefficients? This we will talk about later. So, in the domain if you form the whole matrix this is how it looks like ok. And right hand side is 0. So, this essentially leads to my a, this is my x and this is my b. So, that is the linear system that I am supposed to solve. So, what your numerical technique has done? It has brought down to your physical problem to it is linear system.

Now, how you do that? That process is called the different different techniques finite difference finite element finite volume which brings down the physical governing equations to the linear system. Solution up to this linear system will be the solution of my physical problem, ok.

(Refer Slide Time: 29:27)



## Discretization of NS Eqn

➤ FV Discretization of Incompressible N-S Equation

$$Mu_h = 0$$

$$\Omega \frac{du_h}{dt} + C(u_h)u_h + Du_h - Mq_h = 0$$

Unsteady   Convection   Diffusion   Source

Fluid Flow system.

➤ Time Discretization

$$\frac{du_h^{n-1}}{dt} = \begin{cases} f(u_h^n) & \text{Explicit} \\ f(u_h^n, u_h^{n-1}) & \text{Implicit} \end{cases}$$

INDIAN INSTITUTE OF TECHNOLOGY KANPUR          *Ashoke De*  31

Now when you look at this finite volume discretization of incompressible navier stokes equation, this is in a more generic form one can write; this is the unsteady term convection term diffusion term source term. And the time discretizations for this unsteady term. One can have a explicit scheme, all these details we will discuss later on.

Because we have to and if you go back to the outline of this particular lecture, see that what we talk about first we deal with the diffusion term. So, we will deal with this them individually, then we will deal with the convection term, and then along with the steady and unsteady and finally, with the source term. So, that will build the complete fluid flow system. So, objective of this lecture at the end of the lecture series would be I should be able to solve a fluid flow problem.
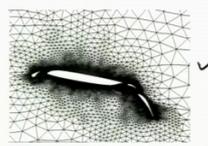
(Refer Slide Time: 30:31)



Now, grids; when we are talking about this numerical discretizations and all this things here one of the essential component is grid.

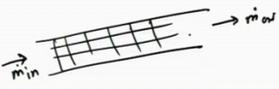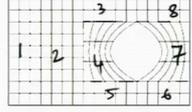(Refer Slide Time: 30:36)



Now, we have used this terminology now I will get you a certain idea. So, this is how the domain looks like. I have a let us say channel, this is my physical problem, there is a flow coming in and there is a few going out. And I have to solve a problem in a channel. So, the problem in a channel would be represented by the governing equations of continuity, momentum and if there is a energy, then energy.

Now, to solve this problem I have to discretized this channel in multiple blocks like this. So, this could be wavy for a geometry which is in curvature in natures, like if you look at this geometry around aerofoil and if you look at the around this particular elliptical shape. Structure grid means all these individual elements are nicely connected with each other and the structure grid very easy to generate over a simple domain. But I am not saying that it cannot be generated over complicated domain, but this is more easy this is called the structure grid where individual elements are nicely connected with each other. So, the indexing if you go by let us say i, you go by j.

So, this indexings are also smooth. And you can have a nice count over this indexing, then this is in (Refer Time: 32:05) getting a structure grid. Unstructured grids these are all unstructured elements. So, advantages of these unstructured grids are for any complicated geometry like aerofoils which are having multiple elements, you can generate unstructured grid very easily. So, you can use over a irregular use a irregular data structure to generate this unstructured grid. So, the advantage over that you can generate over any complicated geometry, but the data structure would be extensive to keep a tag on different different elements.

And block structured grid is that the whole domain is divided in multiple blocks. This is block 1, this could be block 2, this is block 3, this is block 4, this is block 5, 6, 7, 8 like that. The advantage of block structured grid is that if you have a complicated geometry, you can divide them to individual blocks and can generate the grids over that. So, this is how different grids should look like. The structured grid, unstructured grid block structured grid, ok.

So, this is when you talk about the different geometries and you come down to different grids. And this is what you require to solve a realistic problem you have to generate either of different kind of grids. And what you need that your data structures should be able to handle that. So, that is gives you an idea what kind of grids and all this required. So, we will stop here today, and we will take from here in the follow up lectures.

Thank you.