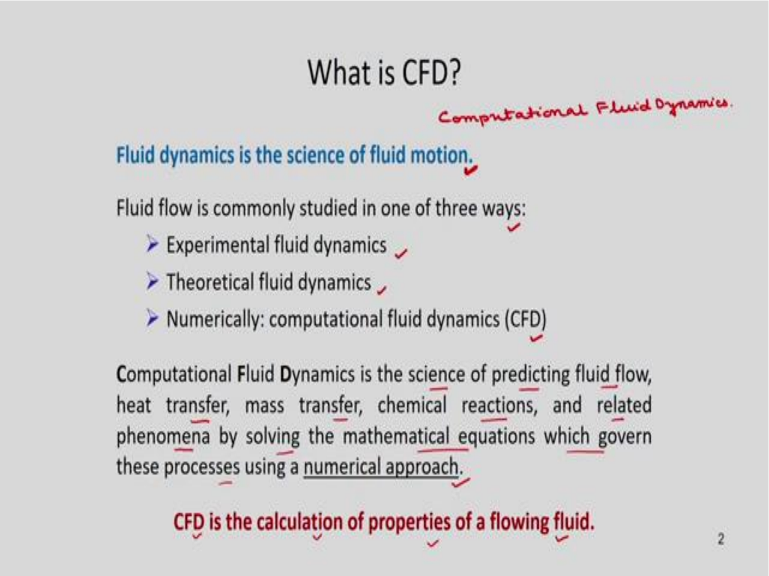
Computational Fluid Dynamics for Incompressible Flows
Professor Amaresh Dalal
Department of Mechanical Engineering
Indian Institute of Technology, Guwahati
Module 01: Introduction to Computational Fluid Dynamics
Lecture 01: Application of CFD

Hello everyone. So, today we will study the application of CFD. So, it is a introductory lecture. CFD is the short form of Computational Fluid Dynamics.

(Refer Slide Time: 00:45)



So, fluid dynamics you know that when we study the fluid in motion then that is known as fluid mechanics. So, fluid dynamics is the science of fluid motion. And CFD is the, this fluid dynamics when we study numerically then that is your CFD, Computational Fluid Dynamics, Computational Fluid Dynamics. Fluid flow is commonly studied in one of the three ways, Experimental fluid dynamics, theoretical fluid dynamics and computational Fluid Dynamics.

So, first one is experimental fluid dynamics, so you can study the fluid dynamics doing some experiments, so you can measure the temperature with thermocouple or the velocity with pitot tube and you can study what is the velocity profile or the temperature profile using experimental techniques.

Other way is that theoretical fluid mechanics or fluid dynamics, you have the governing partial differential equations, which represent the fluid flow, heat transfer or other multi physics problems. So, when you write the partial differential equations, we with some assumptions you can write it to ordinary differential equations and with the boundary conditions if you can solve those equations, then you can have the exact solution.

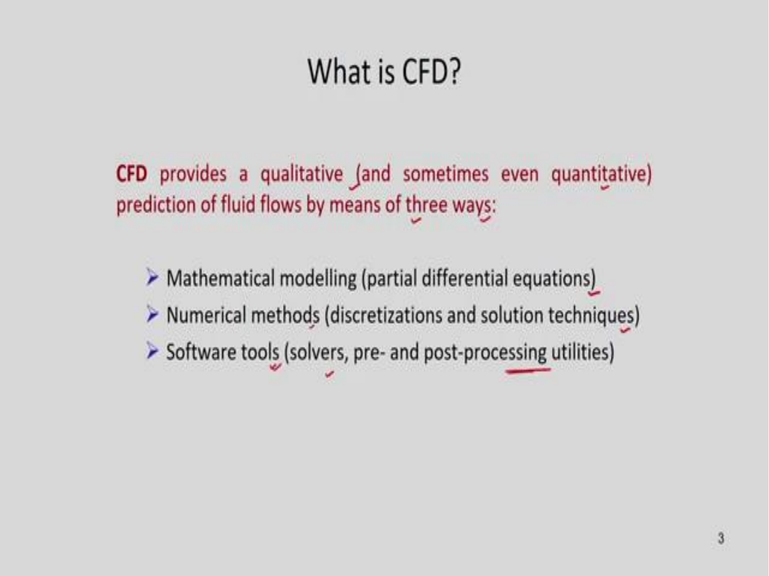
So, theoretical also you can study the fluid dynamics, but you can study theoretical under certain assumptions, because you have to bring down this partial differential equation to ordinary differential equation. Like if you are studying the fully developed fluid flow problem, then fluid fully developed means there is no variation of velocity in axial direction.

So, it essentially boils down to one dimensional problem. And with boundary conditions you can solve and get the velocity profile theoretically. The other way is to study this fluid dynamics numerically. So, you have the partial differential equations using these computational fluid dynamics techniques, you can solve the partial differential equation and get the solution using computational fluid dynamics.

So, computational fluid dynamics is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions and related transport phenomena by solving the mathematical equations which govern these processes using numerical approach. So, you can see that CFD is the calculation of properties of a flowing fluid.

So, it is not only limited to fluid dynamics but you can extend it to heat transfer, you can mass transfer, then some other multi physics problems, electro hydro dynamics flow or magneto hydro dynamics flows or with chemical reaction and many more.

(Refer Slide Time: 04:26)



CFD provides a qualitative and sometimes even quantitative predication of fluid flow by means of three ways, first one is mathematical modelling, then numerical methods, then software tools. So, if you want to solve some problem, then first you find or you write the partial differential equations which govern the flow, so that is your mathematical modelling, then you can choose suitable numerical methods, like discretization techniques and some solution techniques you can choose.

Then you can use some software tools like you can solve using some solvers those discretize equations and get some solutions in terms of data, so only numbers will be there as output. So, to visualize this data you can use some post-processor and in the pre-processor you need to give required boundary conditions and initial conditions if required. So, in post processing tools you will be able to visualize those data and you can see the velocity profile or temperature profile or some contours using some post-processing tool.

(Refer Slide Time: 05:50)

So, we will now see the history of this CFD. So, you can see that earlier CFD work started in 1910. And Richardson used human computer to solve Laplace equation using finite difference method and he solved flow over cylinder only for the potential flow. For inviscid flow he solved using finite difference method. So, in 1910 using human computer.

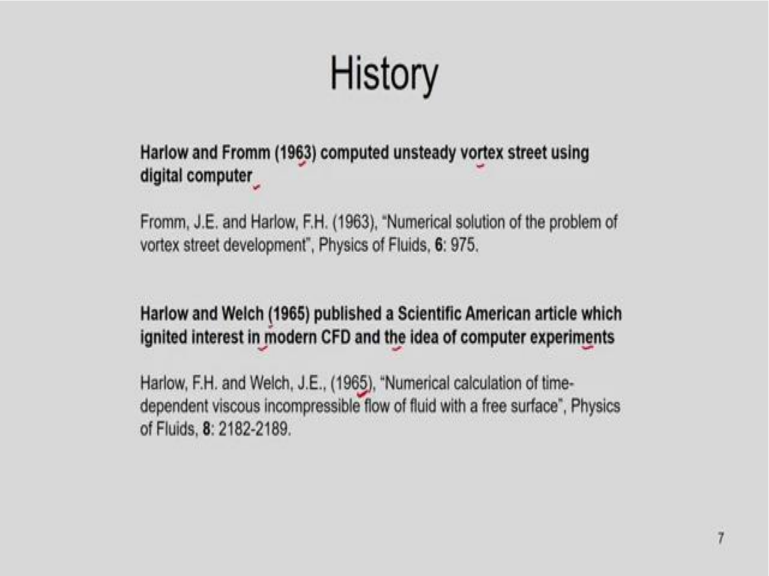
(Refer Slide Time: 06:24)

Then Courant and his research groups like students, Friedrich and Lewy they solved the hyperbolic equations in 1928.

(Refer Slide Time: 06:35)

Von Neumann in 1950 proposed the stability criteria for parabolic problems, which is known as Von Neumann stability analysis and in this course we will study also this stability analysis, Von Neumann stability analysis.

(Refer Slide Time: 06:53)



Harlow and Fromm computed unsteady vortex street using digital computer in 1963. And later Harlow and Welch published a scientific American article which ignited interest in modern CFD and the idea of computer experiments. So, they solved first time the free surface flow, which is a two component fluid flow problems in 1965 using digital computer.

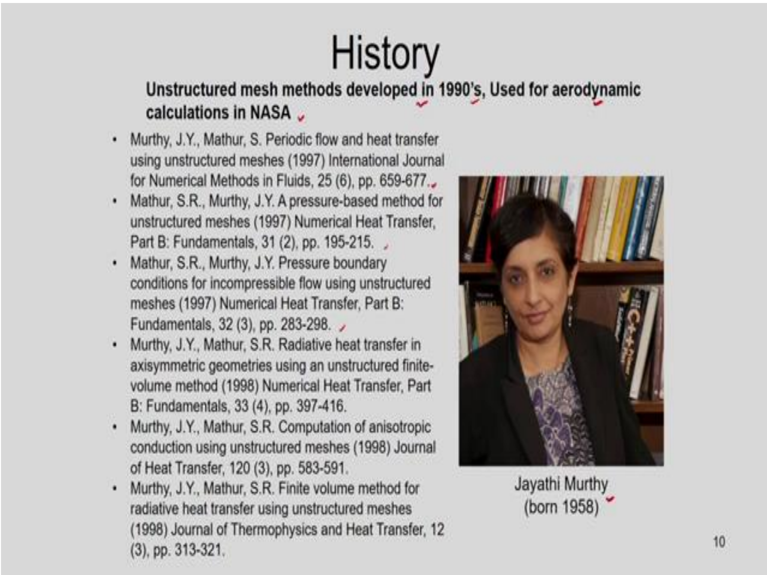
(Refer Slide Time: 07:26)

Professor Brian Spalding is known as the founding father of CFD. So, he and his research group developed this boundary layer codes which is known as GENMIX and in they developed these codes in the years 1960 to 1970s and 1972, they actually those code they put as a software as known as GENMIX. And Patankar is student of Professor Spalding and they, their solution techniques for incompressible flows is published through the 1960s and they first time proposed the simple family of algorithms, which we will study in this course for solving the full Navier-Stokes equations.

(Refer Slide Time: 08:15)

Jameson computed Euler flow over complete aircraft, so this is for compressible flows Euler equation they solved and they published almost 14 papers in 1981 on this using the Euler flow solver.

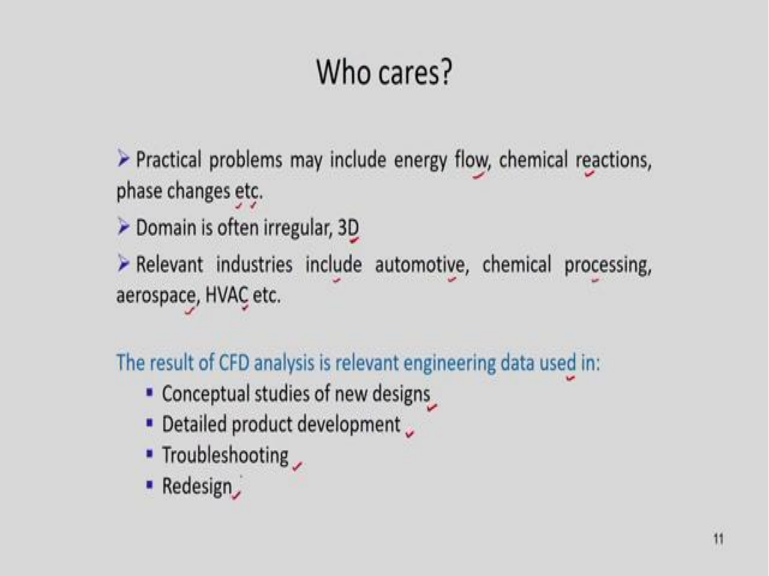
(Refer Slide Time: 08:37)



Later this there was a problem because in the structured mesh it is very difficult to generate for a complicated geometry, so the unstructured grid you can easily fit in a complex geometry. So, unstructured mesh methods developed in 1990s used for first used for aerodynamic calculation in NASA. So, later professor Murthy and her group developed this unstructured grid solvent for ANSYS Fluent. And she and her co-workers developed different modules in fluent.

And you can see the publications during the development and now this fluent is known as ANSYS Fluent and most of you I think you use this commercial structured ANSYS Fluent for solving any problem numerically.

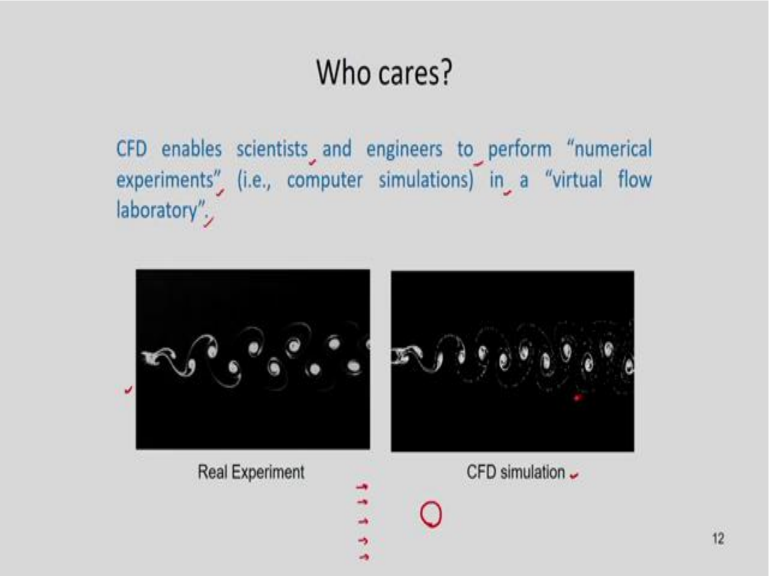
(Refer Slide Time: 09:37)



Now, who are interested in CFD? So, practical problems may include multi-physics like energy flow, chemical reaction, phase change et cetera. And that domain or computational domain mostly these are three dimensional and very complicated. So, it is relevant and so it is easier to use the numerical technique to solve the governing equations.

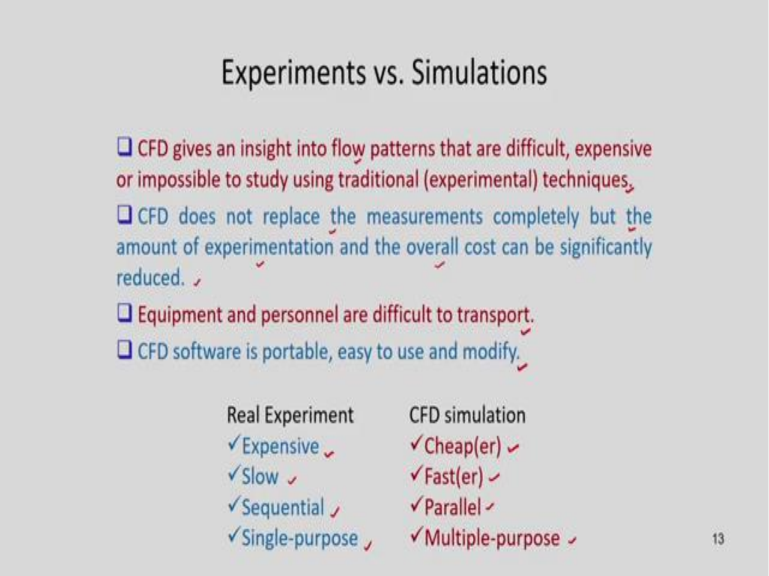
So, relevant industries include automotive, chemical processing, aerospace, heating ventilation and AV conditioner; even nowadays in biomedical applications we use CFD. So, the results of CFD analysis is relevant engineering data used in conceptual studies of new designs, detailed product development, troubleshooting and redesign.

(Refer Slide Time: 10:40)



CFD enables scientist and engineers to perform numerical experiments in a virtual laboratory. So, you can see this is a solution for flow over a circular cylinder, you have a circular cylinder and you have a fluid flow over it, so if you do the real experiment the you can see the visualization. Now, if you do the CFD simulations, you will get this type of simulation result. So, you can see that you are doing actually numerical experiments in a virtual flow laboratory.

(Refer Slide Time: 11:25)



CFD gives an insight into flow pattern that are difficult, expensive or impossible to study using traditional experimental techniques. In experiment, it is very difficult to get the velocity profile or temperature profile at desired or different locations. But, when you use this CFD technique, you discretise these governing equations at a discrete points. So, you can get easily any value, velocity, temperature or species at those discrete point.

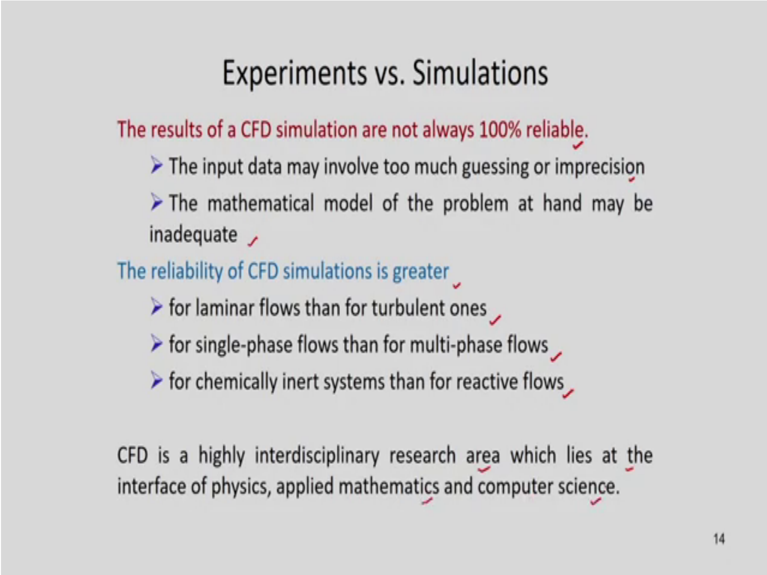
So, it is easy to visualize the results in terms of velocity contours or velocity vector or temperature profile. CFD does not replace the measurements completely, but the amount of experimentation and the overall cost can be significantly reduced. The doing experiment is costly, because you need to fabricate the setup and also you need different instrument to measure velocity, temperature; so it is very costly. So easily you can use this numerical techniques to solve those partial differential equations for a particular problem and get the solutions.

And equipment and personnel difficult to transport and CFD software is portable, easy to use and modify. So, we can see that real experiments are expensive whereas CFD simulations are cheaper; real experiments are slow, CFD simulations are faster; real experiment is sequential, CFD simulations are parallel; real experiments are single purpose, CFD simulation are multiple purpose. What does it mean? That, when you are doing some experiment, so you have made the setup, so you are doing the experiment in the laboratory. So, at a time you can do only single experiments and obviously doing the experiment, fabricating the setup it is very expensive.

Whereas if you develop a numerical solver for solving that problems, fluid flow or heat transfer problems, then once the code is ready you can solve the problem for different conditions parallely. So, that means you can run that solver in different computers for different parameters.

So, parallelly you can run and obviously you can see that it is very portable because you can take the solver with you to somewhere else, but it is very difficult to shift the experimental setup from one location to other.

(Refer Slide Time: 14:40)



The result of CFD simulations are not always 100 percent reliable. The input data may involve too much guessing or imprecision. The mathematical model of the problem at hand may be inadequate. The reliability of the CFD simulations is greater for laminar flows than for turbulent ones. For single-phase flows than for multi-phase flows and chemically inert system than for reactive system.

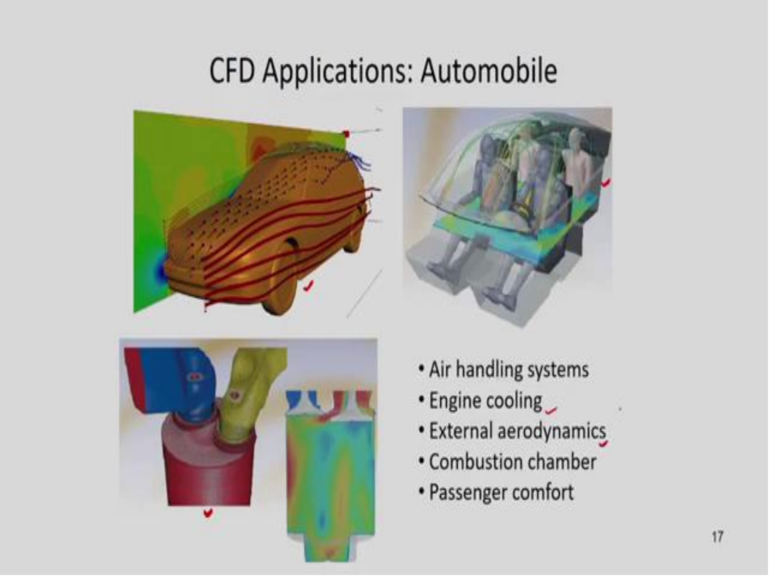
When we solve these equations, we have some assumptions, so obviously, when it becomes more multi-physics then you have lesser reliability. CFD is a highly interdisciplinary research area, which lies at the interface of physics, applied mathematics and computer science.

(Refer Slide Time: 15:29)

Now, let us see few examples or applications of CFD applications, sorry, applications of CFD. So, first, we see in the Aerospace. So, we can see that when you are actually designing this aeroplane, obviously you can solve the governing equations and you can design such a way that you can have the minimum drag while flying.

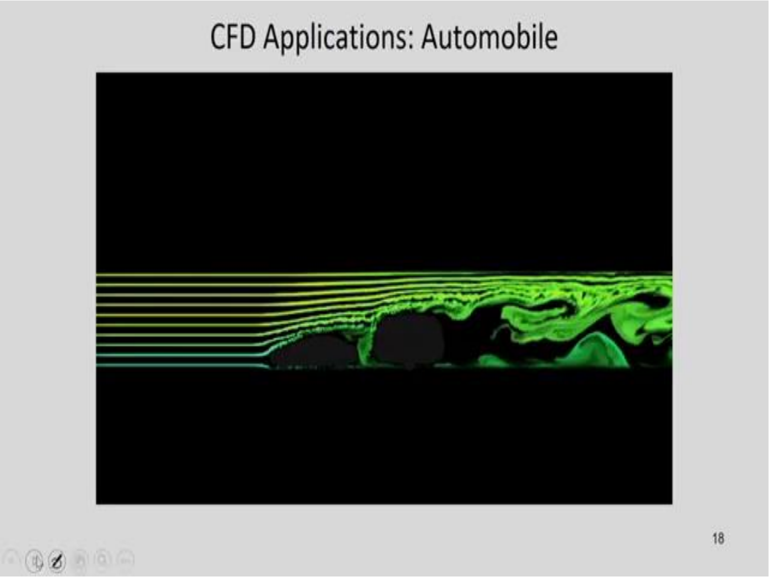
So, obviously, in designing the exterior even for interior design also, you need the safe dissimulations for the passenger comfort, to design the combustor then pumps, missile systems. So, you can use this CFD to design these external aerodynamics, propulsion, missile systems and pumps.

(Refer Slide Time: 16:31)



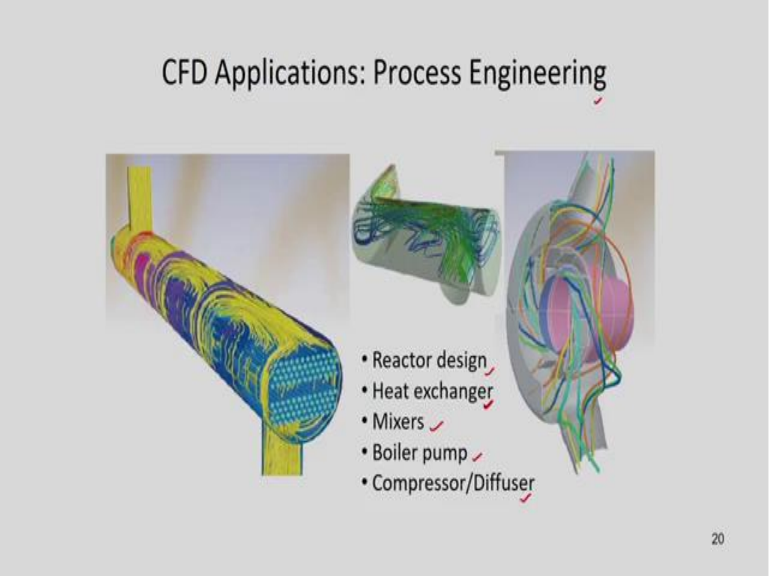
CFD application in automobile, so obviously, here also you can have the you can use CFD for exterior design to reduce the drag, even for the interior design you can use CFD so that the air form the air conditioning reaches to all the passengers to fill the comfort. So, for interior design you can use the CFD, even in the combustion chamber you can use the CFD techniques and also for engine cooling and external aerodynamics you can use the CFD application in automobile.

(Refer Slide Time: 17:14)



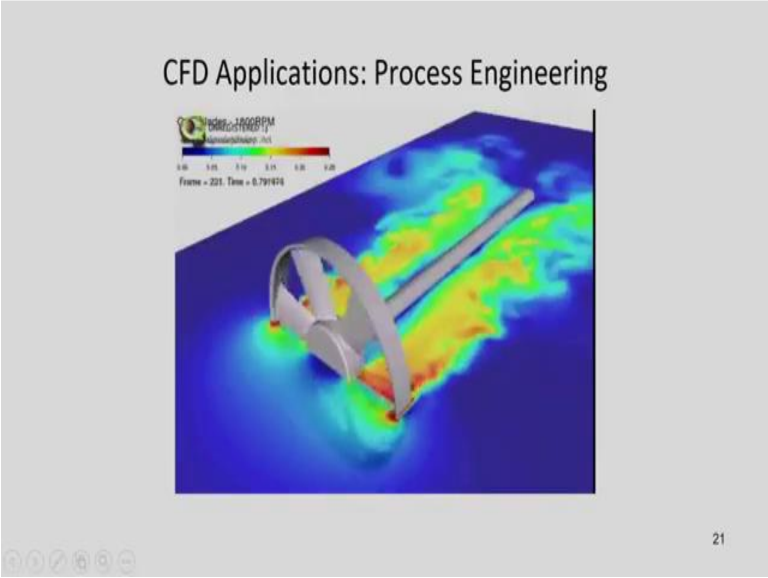
So, you can see some in this animations one car is there and you can see how the flow physics loose behind this car. So, it is very complex you can see, so it is a numerical simulation.

(Refer Slide Time: 17:35)



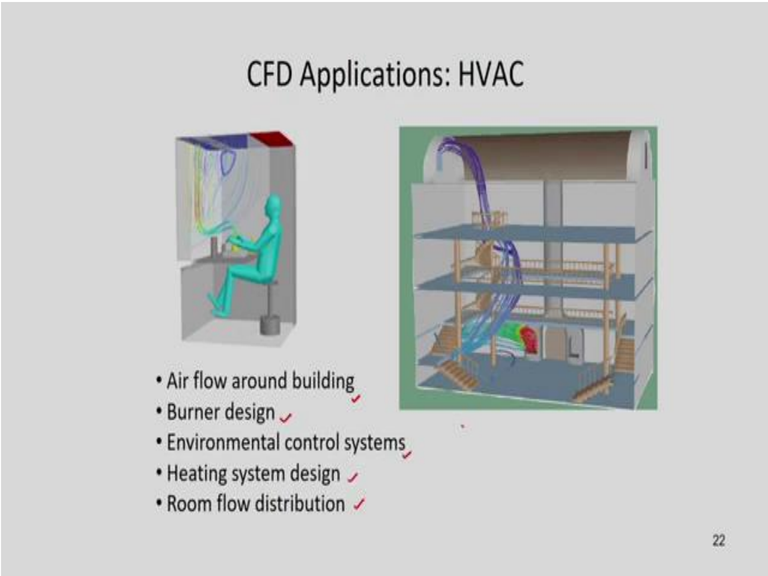
So, some CFD application in process engineering. So, you can use it for reactor design, heat exchanger, so here fluid flow and heat transfer you can solve; mixtures, boiler pumps, compressor and diffuser design you can use this CFD.

(Refer Slide Time: 17:52)



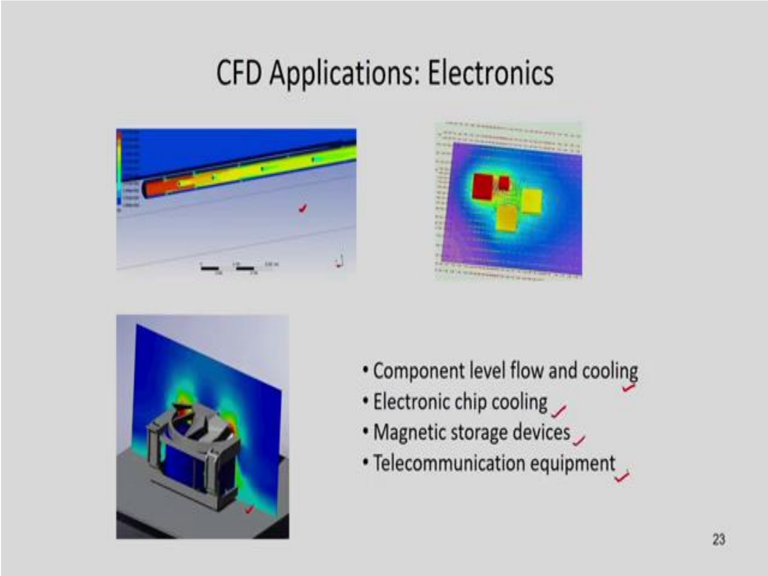
This is some fun is moving and you can see how the flow looks like.

(Refer Slide Time: 18:04)



CFD also you can apply in heating, ventilation and air conditioning. So, when you are designing some room for comfort stay, you can use the CFD technique. So, air flow around building, burner design, environmental control system, heating system design, room flow distributions, all this you can use, to solve this problem you can use CFD.

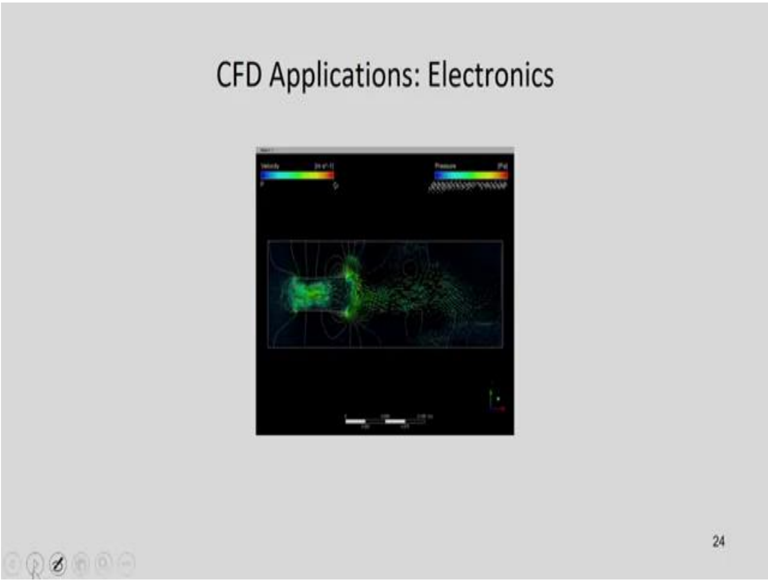
(Refer Slide Time: 18:36)



CFD is also having applications in electronics cooling, so you can see that when you use any computer or laptop you have the processor and if processor is having high temperature, so earlier days in your desktop you will find a fun mounted on the processor and it is cooling. So, these kind things you can actually solve using CFD, you can see this picture where you can see the temperature distribution.

So, this is the fun and here you have the processor with fin mounted on it, so the cooling is taking place due to the force convection, here you can see it is the application in heat pipe, so you can do the CFD analysis for component level flow and cooling, electronic chip cooling, magnetic storage devices, telecommunication equipment.

(Refer Slide Time: 19:39)

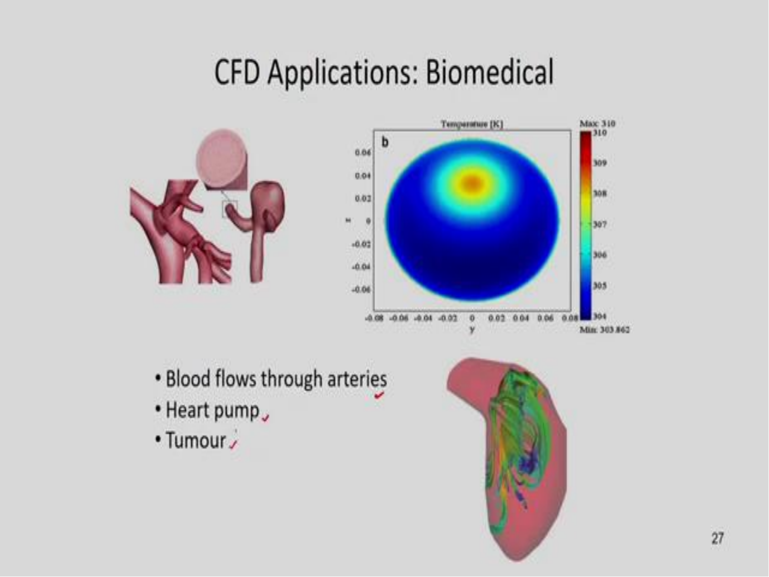


So, some piezo electric device you can see how the fluid flow is taking place, vortex are generated.

(Refer Slide Time: 19:54)

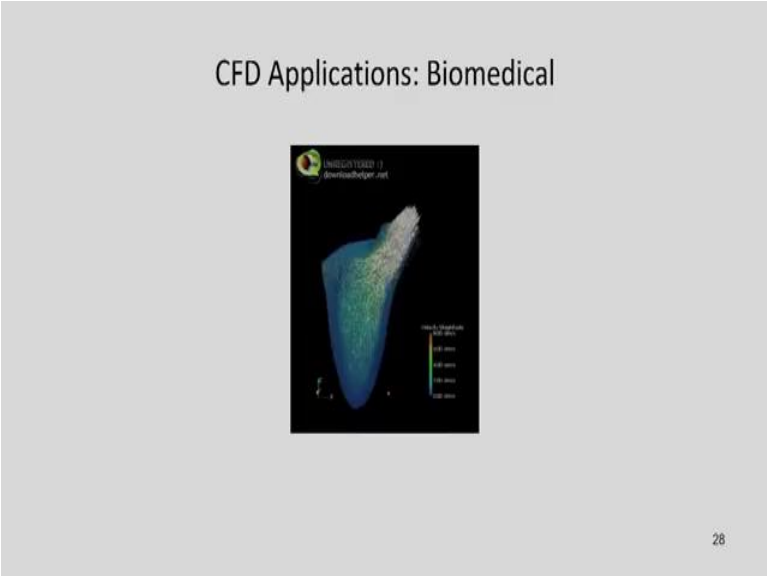
You can also use CFD in sports, so you can see when the ball is moving how the flow physics looks behind the ball you can see from here or if you are cycling then how the flow physics behind you, you can solve using the CFD technique, even for car racing and swimming you can use CFD techniques.

(Refer Slide Time: 20:22)



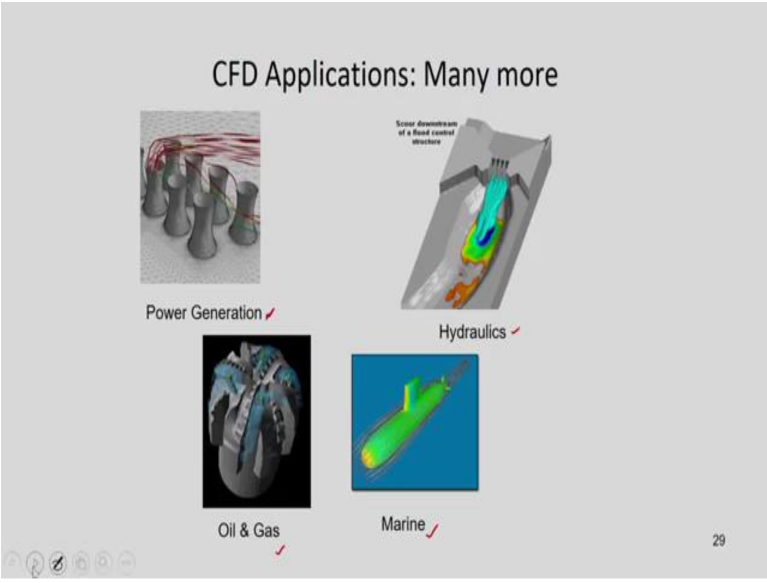
Now, there are many applications in biomedical, so blood flows through arteries, so we can have the deformable arteries as well using different advance technique you can simulate this problems. Heart pumps where you have a moving boundary problems and for tumours you can have the solution of the heat generation inside the tumour and that you can model using CFD.

(Refer Slide Time: 20:58)



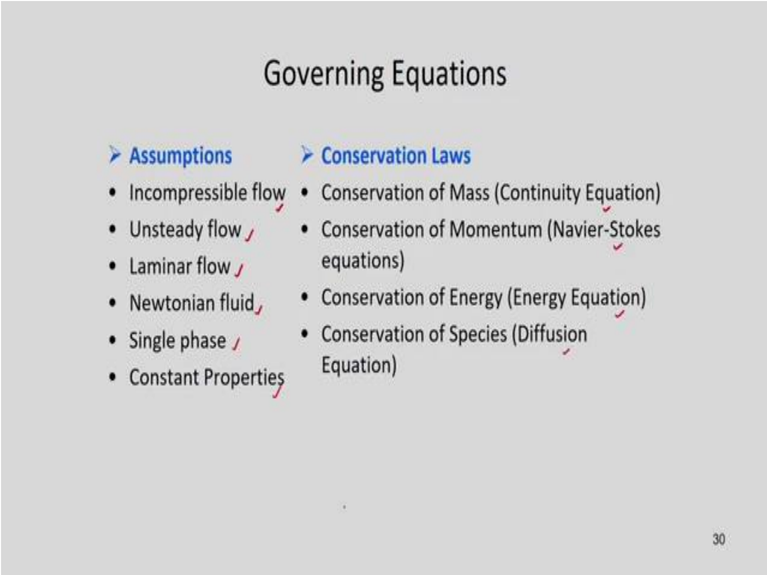
You can see this is the blood flow inside the heart, so one-simulation results.

(Refer Slide Time: 21:07)



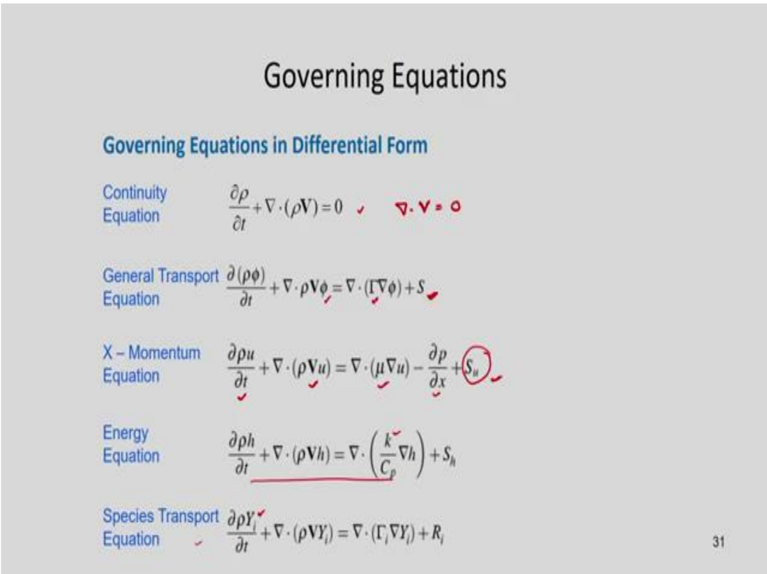
So, there are many applications of CFD in many different industrial in industry and you can see in that power generations, power pants you can have the application of CFD; in hydraulics, so hydraulic turbine all those things you can model using CFD; oil and gas industries, in marine industries, so there are many applications in different kind of industries.

(Refer Slide Time: 21:39)



So, the governing equations you can write; with certain assumptions you can apply this governing equation to some problem. So, asssumptions may be like incompressible flow, unsteady flow, laminar flow, Newtonian fluids, single phase, constant properties. So, for this you can write the governing equations. So, obviously, all the equations you can have the conservation loss, conservation of mass where continuity equation you can write, conservation of momentum where Navier-Stokes equation you will write; conservation of energy, energy equation; conservation of species, diffusion equation.

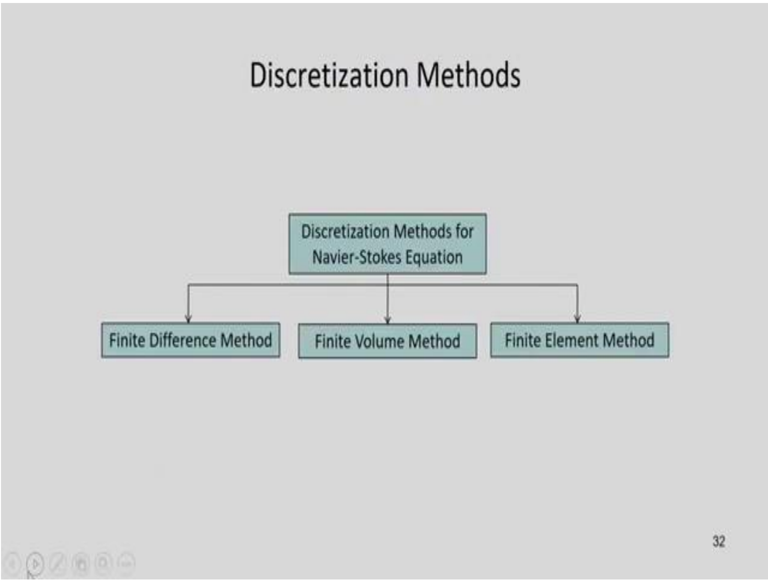
(Refer Slide Time: 22:15)



So, you can see that this is your continuity equation in general but if it is incompressible flow, obviously it will be diversions of v will be 0. Then, you have general transport equations; for any species you can write this equation, especially for s momentum equation if you write this equation then it is the temporal term, this is your convective term and this is your viscus term, this is the pressure gradient term and if you have any source term that you can write, energy equations in terms of enthalpy if you write, then this is the energy equation where k is the thermal conductivity.

Species transport equation in terms mass fraction, Y i is the mass fraction, so that you can write, so you can see you can see all these equations you can write in a general transport equation for a general variable, phi where s may be different or the diffusion coefficient gamma will be different.

(Refer Slide Time: 23:17)

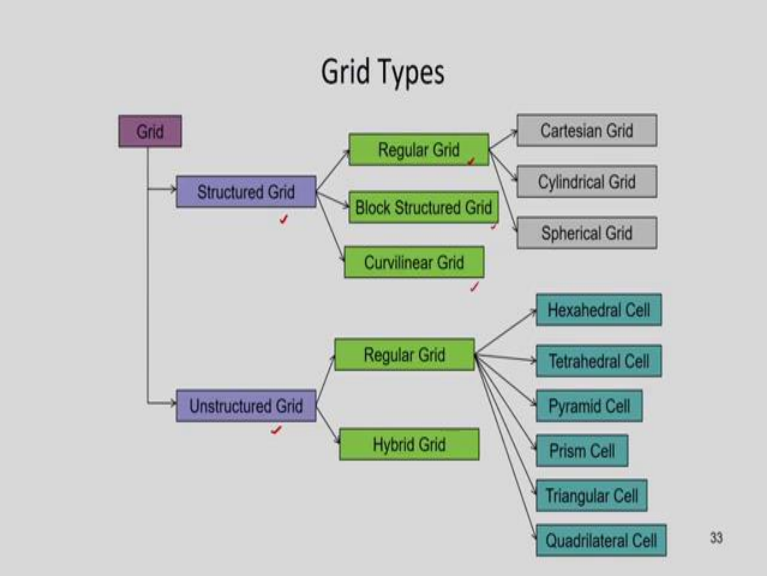


So, when you have the governing equation you need to use some discretization techniques, so mainly in CFD we use three different techniques, one is finite difference method, then finite volume method and finite element method. In this course, we will study only finite difference method and finite volume method.

Finite difference method, generally we use Taylor series expansion and find the approximation of any derivative, first or second derivative and we discretize the governing equations and write the final algebraic equations. When we use finite volume method then we integrate the governing equation over a control volume and write the discretize equation.

But, in finite element method we integrate the governing equation with some waiting function in a particular element and we write the discretise equations. So, obviously, in this course we will study only finite difference method and finite volume method.

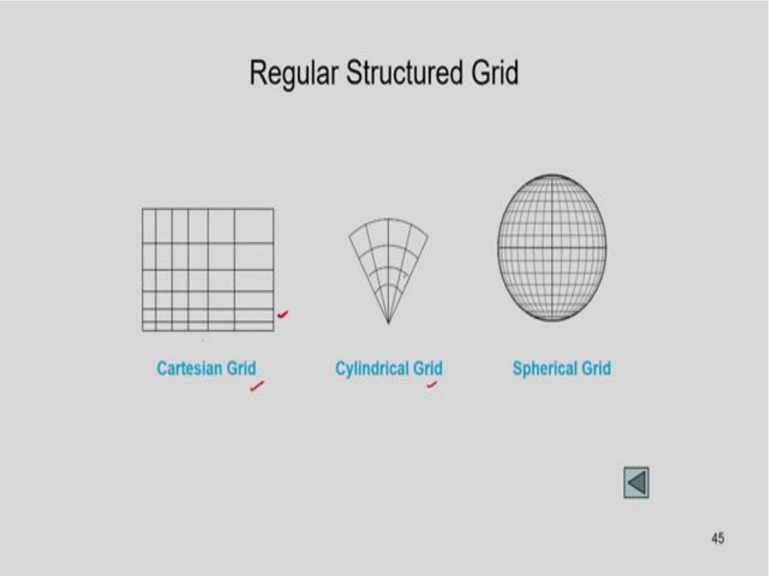
(Refer Slide Time: 24:19)



To solve these governing equations you need to discretise this domain into grid. At those discrete point, you need to solve the discretised algebraic equations. So, now grid can be classified into two; structure grid and unstructured grid. So, as I told before that structure grid are easy to generate in a simple geometry, but if we have a complex geometry then it is very difficult to generate structure grid, so you need to use, divide the domain into blocks, and in the block you need to generate the mesh.

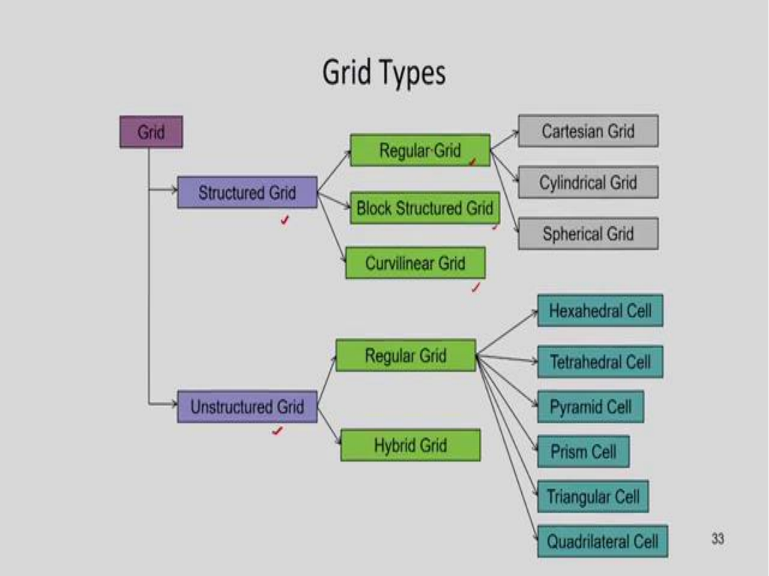
But, unstructured grid is it is very easy to fit into a complex geometry. So, structured grid can be further classified as regular grid, block structured grid and curvilinear grid.

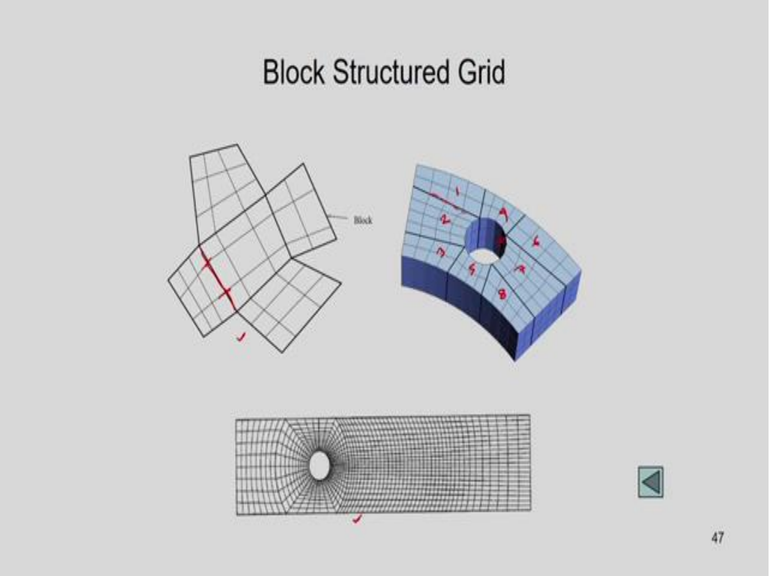
(Refer Slide Time: 25:14)



So, in the regular grid, regular structured grid you can see these are actually orthogonal in the coordinate system. So, if it is we use Cartesian grid then this grids are orthogonal to each other, it can be uniform spacing or non-uniform spacing; you can see here we have used non-uniform spacing. Cylindrical grid, so in cylindrical coordinate system these are orthogonal to each other. And in spherical grid, in spherical coordinated the grids are orthogonal to each other. So, these are regular structured grid.

(Refer Slide Time: 25:47)

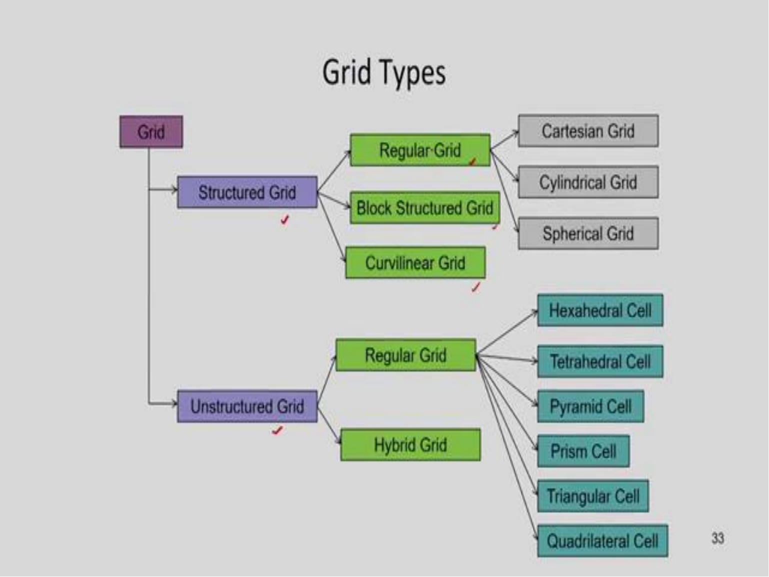


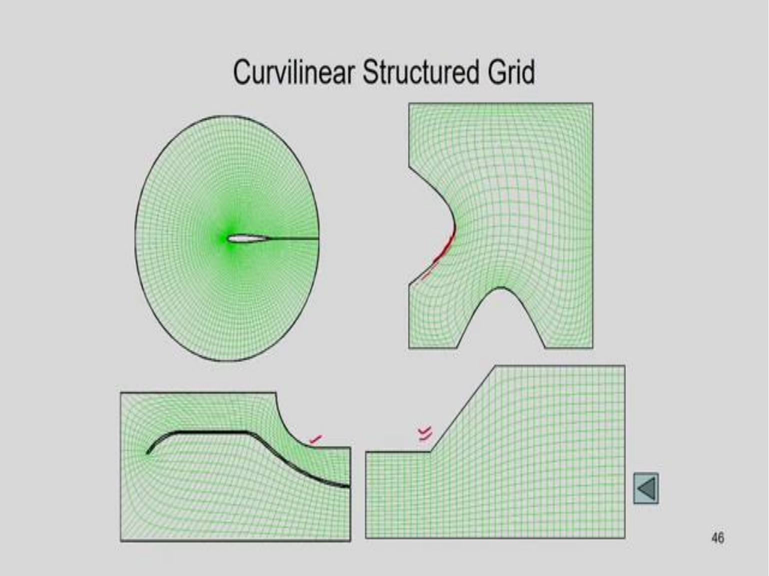


Then block structured grid. So, you have a complex geometry, then you can divide the domain into zones and generate the mesh and at the interface you keep the continuity, make the continuity. So, we can see it is some cylinder is there in this here, so it is cylinder is there. So this is divided into 1, 2, 3, 4, 5, 6, 7, 8, 8 subdomains and each domains the grids are generated.

And it is a continuity is maintained at the interface. So, this is one the representation of the grid for a flow over circular cylinder. So, these are known as block structured grid because you are dividing the domain into blocks and in each block, you are generating the mesh.

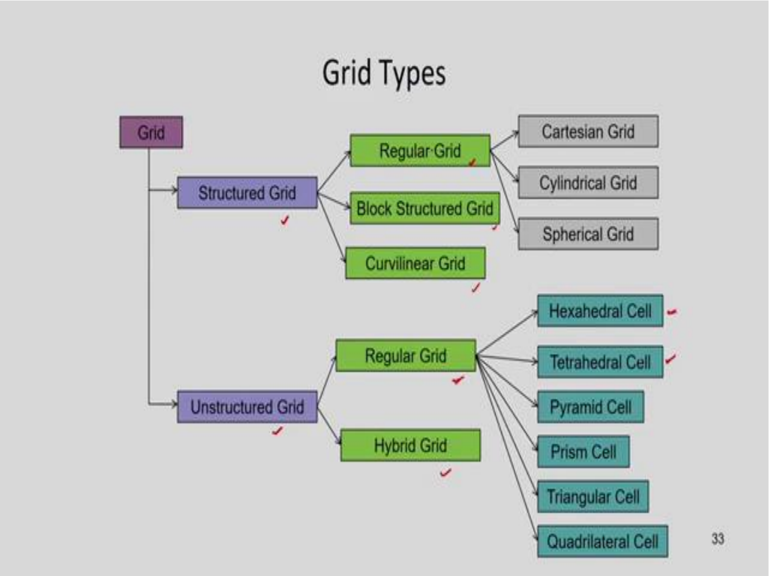
(Refer Slide Time: 26:42)





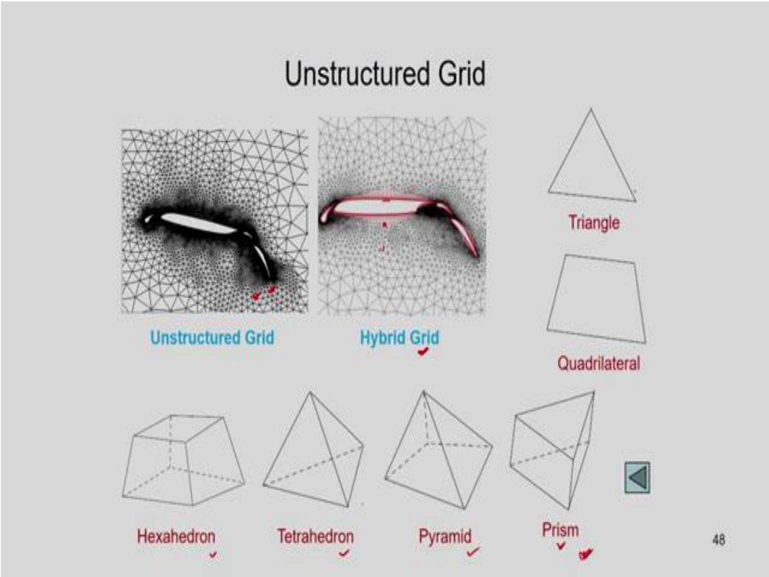
Then curvilinear grid. So, curvilinear grid. So, curvilinear grid are known as body fitted grid, so you can see that the grids are following the body, surface of the body. So, here you can see if it is this is the body the grids this is the grid so it is following the body. So, these are curvilinear structured grid, so obviously, these girds are not orthogonal you can see. So, these are non-orthogonal grids. And for different geometry you can see that how the curvilinear grids are generated. In curvilinear grids, as I told that, grids will follow the boundary.

(Refer Slide Time: 27:17)



In unstructured grid, so we can have regular grid and hybrid grid. In regular grid, you can have same type of cells, like hexahedral, tetrahedral, prism or pyramid. In 3D and triangular cell or quadrilaterals in 2D. And hybrid grids are grids where you can have more than one type of cells; if you have hexahedral cells and tetrahedral cells are mixed then obviously you can have hybrid grid.

(Refer Slide Time: 27:44)



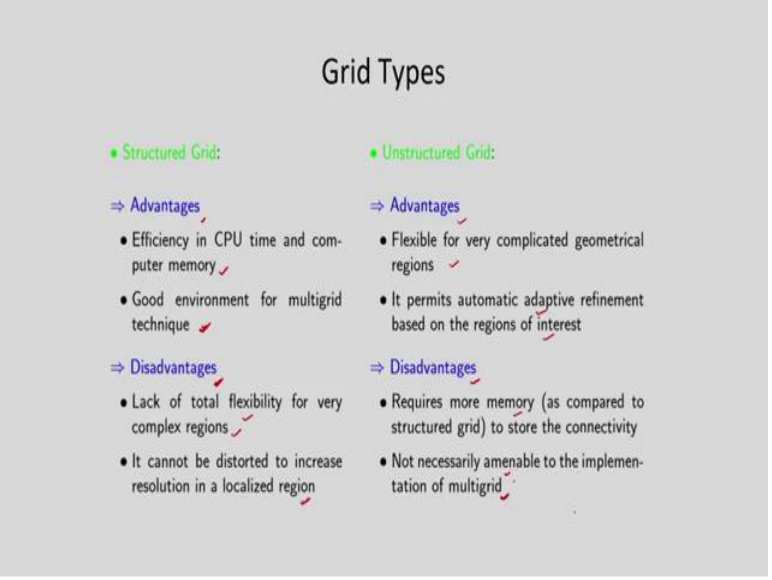
So, you can see here, so you can see that in this case this first figure, you can see the grids are fine near to the boundary to capture the gradient mode correctly or mode accurately. So, local refinement you can do using unstructured grid. But, in structured grid, it is very difficult to do local refinement because to maintain the continuity this will be extended in other directions.

So, in unstructured grid that is why it is very easy to use the local refinement. And you can see you can use hybrid grid, hybrid grid mean more than one type of cells if you use. In this case, you can see near to the boundary you have quadrilateral cells and away from the boundary, you have triangular cells.

So, that means near to the boundary you have almost structure and orthogonal mesh so that you can capture the gradient correctly. And away from the surface where you do not have much gradient you can use triangular cell, so this is kind of hybrid cell. So, in 3D, you can have hexahedron type cells or tetrahedron or pyramid or prism.

Now, it is in commercial softwares they use any type of poly hydra, but Ansys Fluent earlier they used use only these four types of cells for the three dimensional domain. And for surface grid, triangular or quadrilateral mesh are used.

(Refer Slide Time: 29:30)



So, you can see there are some advantages and disadvantages of this structured and unstructured grid. So, you can see the structured grid, what are the advantages, efficiency in CPU time and computer memory, because in structured grid you can easily find the neighbours, because you have the indices I, J, K and I plus 1, I minus 1, or J plus 1, or J minus 1, will give you the neighbours.

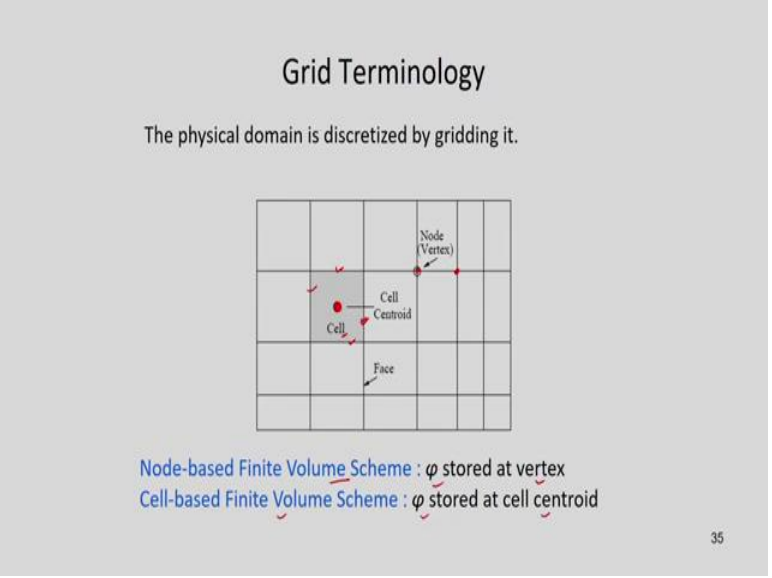
So, less storage memory is required and it is good environment for multigrid technique. Whereas, in the unstructured grid, advantages are flexible for very complex geometry region and it permits automatic adaptive refinement based on regions of interest, so at the particular region if you need very fineness then you can use actually unstructured grid.

The disadvantages of the structured grid is that lack of total flexibility for very complex regions because you cannot do local refinement and it cannot be distorted to increase resolution in a localised region. The unstructured grid disadvantages are there, that requires more memory as compared to structured grid to store the connectivity.

So, in unstructured grid you cannot find the neighbours with the indices I plus1, I minus 1, because there is no structured way these grids are oriented. So, in pre-processing stage, you need to find the neighbours or connectivity so that you can use it later while calculating the fluxes.

So, obviously, you need more memory requirement to store this connectivity and the neighbours' informations. And this another disadvantage for unstructured grid is not necessarily amenable to the implementation of multigrid.

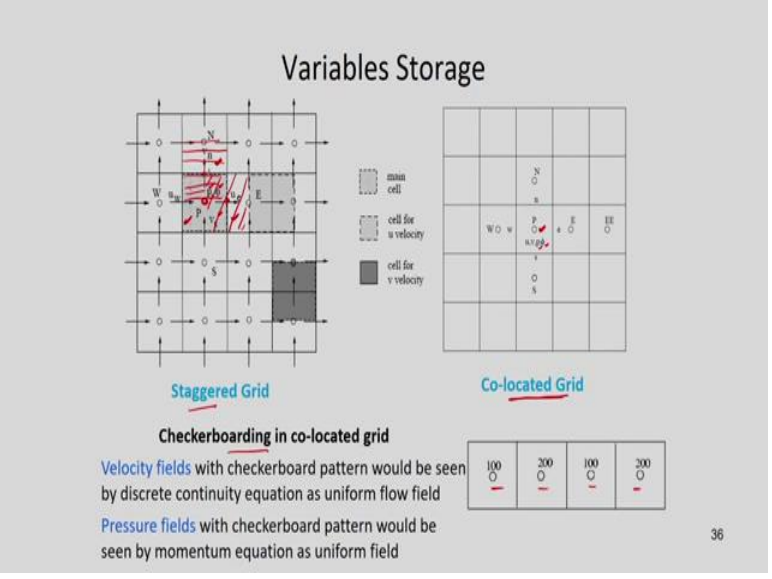
(Refer Slide Time: 31:25)



So, now we will talk about the grid terminology. So, when you discretise the domain for generally, for finite difference method we solve the equations, discretised algebraic equation is solved at this grid node, which are known as vertex.

But, in case of finite volume method we integrate over this cell, so the value is stored at the cell centre or cell centroid and this is the cell and these are known as faces, so in faces in two dimension it is line, but it is three dimensional it will be a surface. So, you can have node based finite volume scheme where phi the variable stored at vertex, or cell based finite volume method where the variable stored at the cell centroid.

(Refer Slide Time: 32:22)



Based on the variable storage you can have two different types of grid arrangement, one is staggered grid and another is co-located grid, we will learn more detail when we will study the finite volume method, but here just I will just introduces that, when all the variables you store at the cell centre or at the same node then it is known as co-located grid. So, you can see the velocities, pressure and any species let us say temperature or any species you can store at the cell centre at the same point then this is known as co-located grid.

But, you can see that it is very easy to have the data structure, because all the variables are stored at the cell centre, but it is having some disadvantages, so that we will learn later, that is known as velocity pressure decoupling. So, if you use co-located grid, it may be possible that your velocity and pressure are not talking each other because your velocity there will be due to the pressure difference there will be velocity.

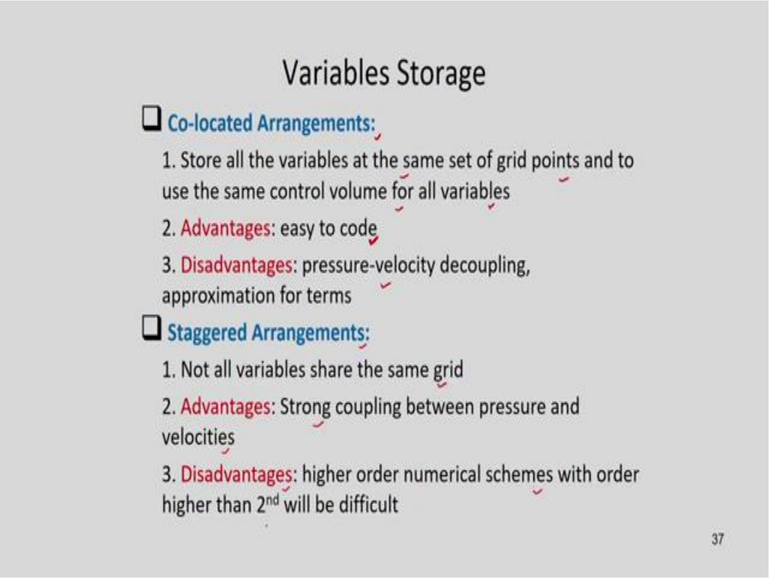
So, if there is no pressure difference then obviously there will be no velocity, so this pressure and velocity if it is decoupled then there will be problem, so that is known as pressure velocity decoupling. And when you have this checker boarding kind of distribution of velocity or pressure then you will get this type of problem in co-located grid. So, that we will discuss in detail later.

And in staggered grid to avoid this problem, velocity pressure recoupling problem, we use staggered grid, where we store the variable at different places. So, we solve the pressure and the scalers like temperature or species at the cell centre and the phase centre in staggered way we will solve the velocities. So, we can see in this figure, so if this is the figure where we are solving at the cell centre only the pressure and any scaler like temperature or species, whereas we solve the velocities at the at this cell.

So, this is velocity, U velocity and at the cell we solve the B velocity. So, you can see that U and B are solved at staggered way and pressure and any scaler are solved at the cell centre of the main control volume. So, this is the main control volume for pressure and scaler but this is the control volume for U velocity and this is the control volume for B velocity.

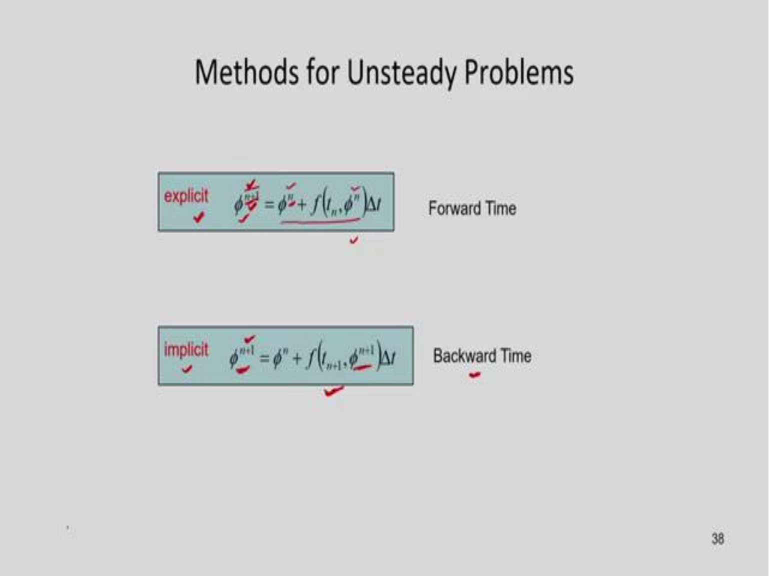
So, in staggered grid, actually you can avoid this pressure velocity decoupling, but you can see that storage requirements are different. Because as and U or B are served and different places, so you need to write the code carefully so that you can take care about the data structure.

(Refer Slide Time: 35:41)



So, you can see that in co-located arrangements store all the variables at the same set of grid points and to use the same control volume for all the variables. Advantage, obviously, it is to code, because all the variables are stored at the same point. Disadvantage is, pressure velocity decoupling. And in staggered arrangement, not all variables share the same grid, so advantage is strong coupling between pressure and velocities. Disadvantage is higher order numerical schemes with order higher than second order will be difficult.

(Refer Slide Time: 36:16)

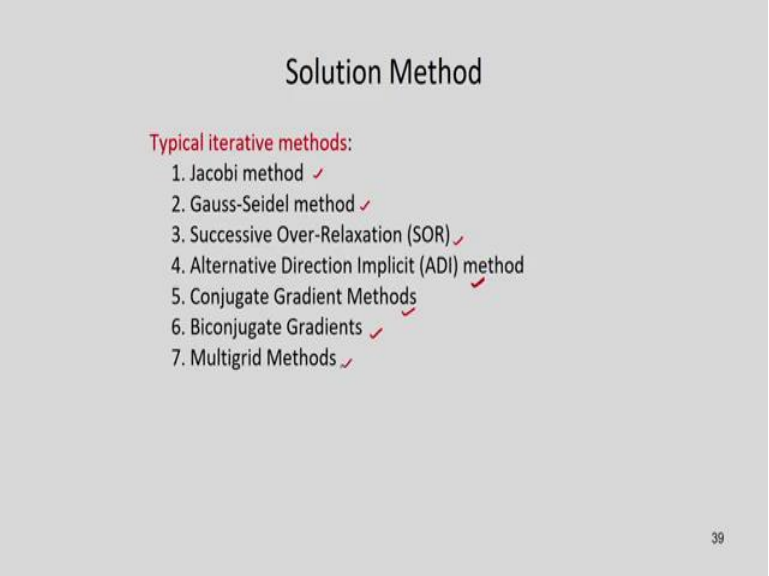


For the unsteady problems, also if you are discretising and using forward time then this is known as explicit method where you have only one unknown, where n plus 1 is the current time step and n is the previous time step. So, this is known as explicit method. So, only one unknown is there and you can have in the right hand side all known terms at the time level n.

But if you use backward time discretization method for the temporal gradient then it is known as implicit method, where you have more than one unknowns. You can see it is n plus 1 and here also n plus 1, so obviously you have more than one unknowns at n plus 1 time level, which is the present time level. So, this is known as implicit method.

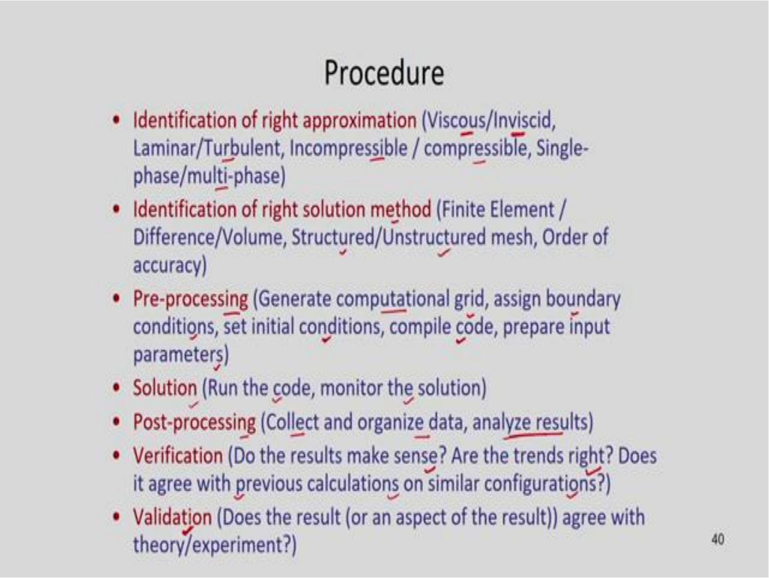
So, in explicit method as it is only one unknown it is easy to solve, but where in implicit method where you have more than unknowns so you have to use some numerical techniques, sorry numerical solvers to solve these equations.

(Refer Slide Time: 37:24)



So, for solving this discretization equation, you can use these iterative methods, one is Jacobi method, other is Gauss Seidel method, we will discuss in detail later, Successive Over-Relaxation, Alternative direction implicit method, so these are some iterative methods. We have some other methods where which are known as conjugate gradient methods, biconjugate gradient methods and you can also use multigrid.

(Refer Slide Time: 37:49)



So, you can see that when you are solving some problems using CFD, first you identify the right approximation and write down the governing equations. So, identification of right approximation means, whether it is Viscous or inviscous inviscid, because you can write the governing equation accordingly, whether laminar or turbulent, incompressible or compressible, single-phase or multi-phase, so accordingly you first identify the right approximation and write the governing equations.

Then you identify the right solution method. So, which discretization scheme you want to use, say finite difference method, finite element method or finite volume method, whether you can use want to use structured or unstructured grid. And what is the order of accuracy you will use, temporal and spatial both. So, accordingly you need to discretise these partial differential equations.

Then in pre-processing stage, generate the computational grid, because at discrete points you need to solve this discretized equation, so depending on your choice whether structured or unstructured grid you generate the computational grid, then in pre-processing stage you assign the boundary conditions. If it is time varying then with solutions you need to apply the boundary conditions.

Then set initial conditions for the unsteady problem. If it is a steady problem then you need to have the guess solution at starting while starting the iterative method. Then once all these are done then you compile the code, so if there are some errors then you fix the bug, then prepare input parameters, then you solve this, run the code, monitor the solutions. So, whether your error is decreasing with iteration or time that you check and monitor.

Once you get the conversions then you get the results. But, result you will get in terms of numbers only. So, at discrete points you will have the values of particular variables. So, those you collect and organize the data, then you use some post processing software and analyse the results.

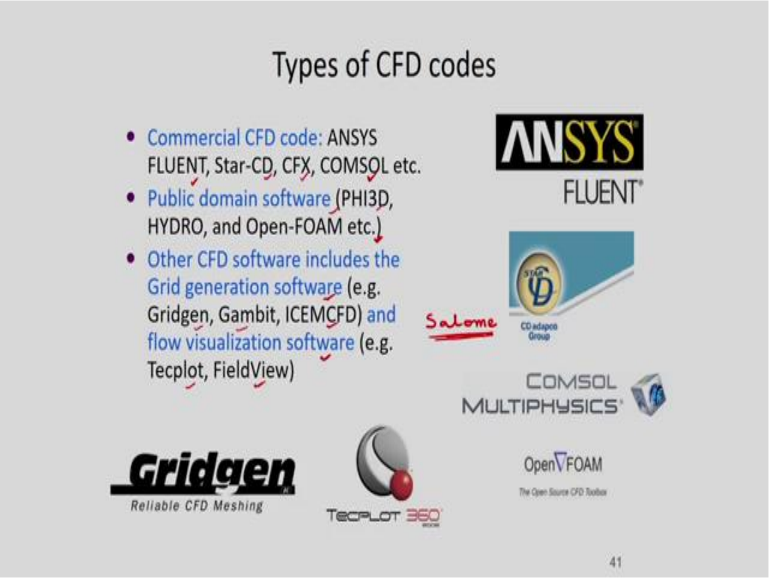
Now, once you post process it, first you need to verify the solution, what is verification? Whether these are physically correct or not. Because if you are solving a fluid flow problem in a channel, let us say the flow is taking place from left to right, but if you are getting the solution from right to left, the velocity is coming from right to left , then obviously it is not physically correct. So, based on your problem whatever you have chosen, so you check whether the results are physically correct or not.

So, do the results make sense? Are the trends right? Does it agree with previous calculation on similar configurations? So, that you verify. Once you get it, then you solve a known problem, which already the solution is available in literature, so that is known as validation. So, when you write the solver first time, you need to know whether the solver is giving correct result or not, so to test it you need to solve a known problem which is available in the literature.

So, does the results or the aspect of the results agree with theory or experiments? So, you can have some numerical or experimental results available in literature. So, qualitatively you can check first whether these are matching or not, but you need to verify also quantitatively. That means you need find the velocity distribution at a particular line or a particular area then you can compare it with the available results in literature, whether it is numerical or experimental but need to compare, or any other values like drag coefficient or lift coefficient or the moment coefficient or the shear stresses, distribution along a wall.

So, all those things quantitatively you need to compare with the with your results and the results available in the literature to verify your solver so that you will be confident that your code is correctly written and there is no bugs.

(Refer Slide Time: 42:55)



So, there are different types of CFD codes available like commercial CFD code ANSYS Fluent, Star CD, CFX, then COMSOL it is a very multi physics software, so ANSYS Fluent, Star CD these are written in using finite volume method, but COMSOL is a written using finite element method. You have some public domain software like PHI3D, HYDRO, Open FOAM. And for this solving the multi-phase flow you have Gerris and Basilisk, so that are also available and these are free, you can download and you can use it.

And other CFD software includes the grid generation software, obviously you need to generate the grid before solving the equations. And there are some software like Gridgen, Gambit, ICEM CFD, these are some commercial softwares available but also some open source software is there like Salome, so you can generate structured and unstructured grid and it is having the graphical user interphase also to generate the grid.

And to post process the results you have flow visualization software, so Tecplot, FieldView, you can use to post process the results and visualize the results, so these are some commercial and open source software we discussed. So, in today's lecture, we have seen why we need to use the computational fluid dynamics and what are the procedures to solve any problem using CFD.

We have also discussed some history of CFD, then we have seen some application of CFD in particular areas and later we have seen the different type of grid, structured grid and unstructured grid; we have also seen different grid arrangement like co-located grid and staggered grid. After that, also, when you discretize the equation, you will get the discretized algebraic equations.

But you need to solve it, so for solving it you need some solver, so we have seen some iterative solvers, like Jacobi, Gauss Seidel or Successive-Over-Relaxation and also some direct method like conjugate gradient method or biconjugate gradient method. Those things we will discuss more detailed in other modules.

And we have seen that when you solve a problem, what are the steps you need to follow for solving any problems. Thank you.