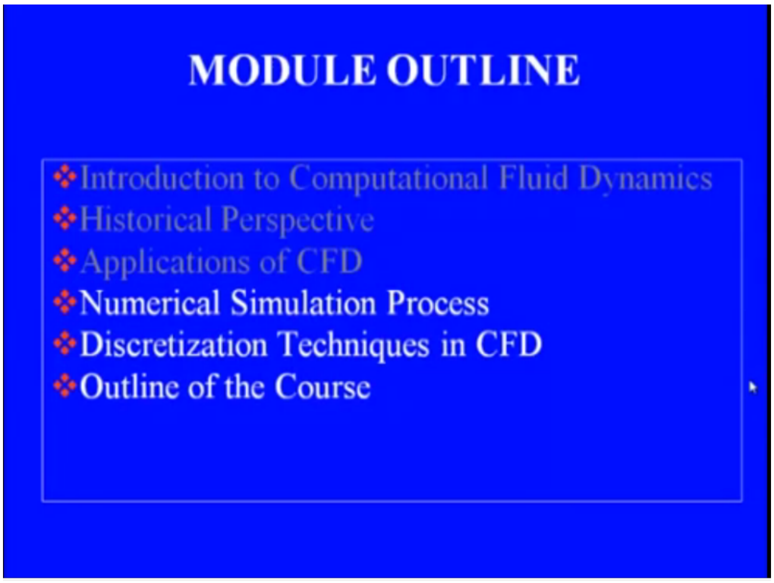
Computational Fluid Dynamics
Dr. Krishna M. Singh
Department of Mechanical and Industrial Engineering
Indian Institute of Technology - Roorkee

Lecture - 02
CFD: Simulation Process and Course Outline

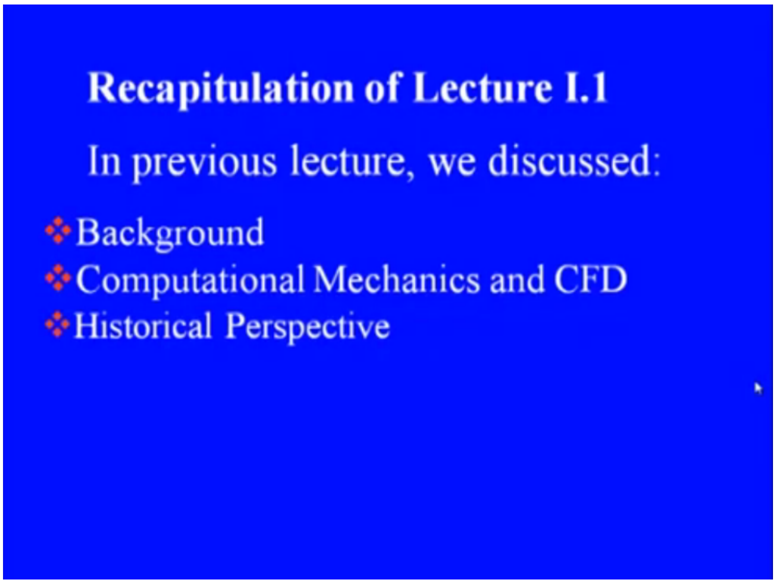
Welcome to the second lecture in module 1 on introduction to computational fluid dynamics. In the previous lecture, we had provided with a brief introduction to computational fluid dynamics. We had a look at the historical perspective and we also had a brief look at myriad applications of CFD.

(Refer Slide Time: 00:47)



Now in today's lecture, we would focus on the basic numerical simulation process and what are the main discretizing techniques which are used in CFD? And we will also briefly discuss about the outline of the course at further topics, which we are going to cover in this introductory course on computational fluid dynamics.

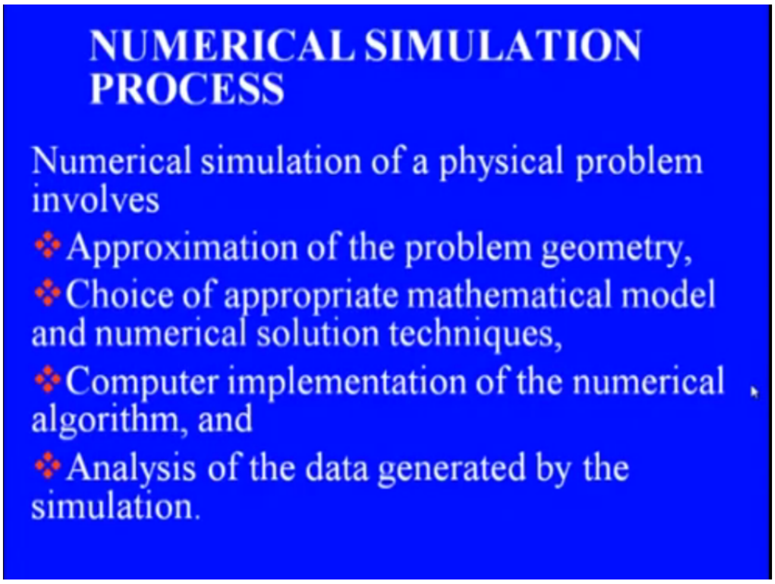
(Refer Slide Time: 01:03)



So let us have a brief recap of previous lecture. We discussed what is meant by computational mechanics and CFD? And what is the need for numerical simulation? We also had a brief historical perspective based on the developments of the algorithms and early applications and what is today happening in CFD research and applications? We also had a look at major applications of CFD.

Now today we will have a brief look at the CFD simulation process and then we will discuss for topics which we are going to cover in this course.

(Refer Slide Time: 01:48)



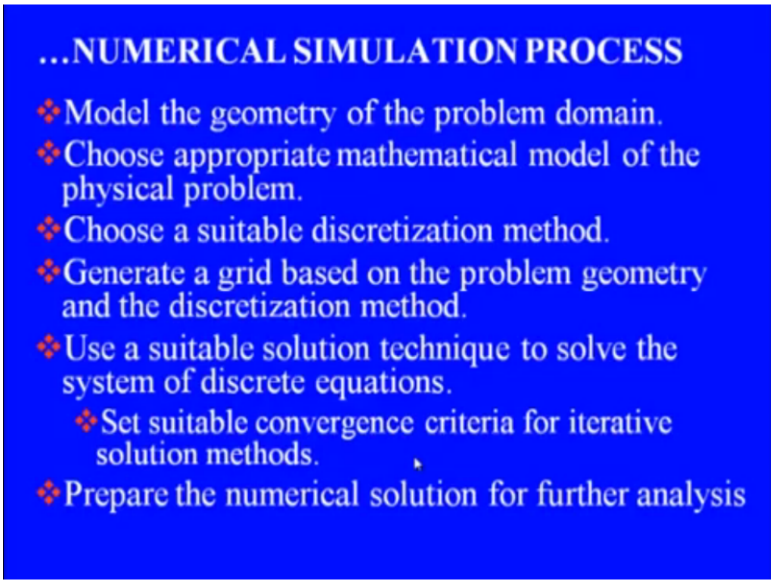
So let us start with numerical simulation process as conceptual outline and then we will briefly discuss the discretizing techniques, which have been used in computational fluid dynamics and then the outline of the course. Now let us have a look at the numerical simulation process. At numerical simulation for any problem what are the conceptual steps are involved which we must address too. So the steps involved are that we will have to first obtain an approximation of the problem geometry.

The CAD model, which might have been made for the production of prototype may not be suitable in its entirety for the CFD analysis and we might have to make certain approximations to make that geometry amenable for a CFD analysis purposes. So this is what we mean by approximation of the problem geometry. Then we have to choose the appropriate mathematical model and numerical solution techniques depending on the physics of the problem and the expertise available are the softwares, which are available to us.

And then once we have made this choice about the mathematical model and numerical simulation techniques, we have to implement it on a computer and then once the numerical simulation technique has been implemented, we have done our problem, we have got huge set of data for flow variables that data has to be analyzed to get meaningful physical insight into the problem.

So these are basic steps which we could be involved in any numerical simulation of a physical problem.

(Refer Slide Time: 03:34)

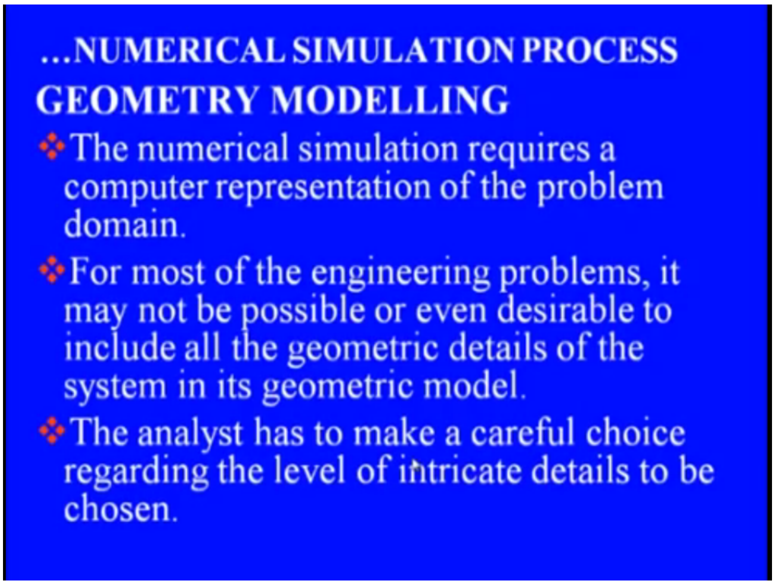


Now let us have a look at each one of these with more detail. The first step involved would be model the geometry of the problem domain and choose appropriate mathematical model and choose a suitable discretizing method, this would be clearer when we discuss about discretizing schemes. We can choose a finite difference, finite volume, finite element method or a meshless scheme.

And then depending on the choice of the discretizing method, we have to generate a suitable grid which would also depend on the problem geometry and the physics of the problem and then we have to make choice of the suitable solution techniques, which can solve the system discrete equations, which we have obtained after the application of the discretizing method on our continuum problem.

And in this step most of the time we would be using iterative solution methods. So we have also got to set suitable convergence criteria that what sort of convergence we require? What is its tolerances we can accept and so on? And first we have obtained the solution, prepare the numerical solution for further analysis. Now let us have a more detailed look at the major steps, which we had discussed in the numerical simulation process.

(Refer Slide Time: 05:04)

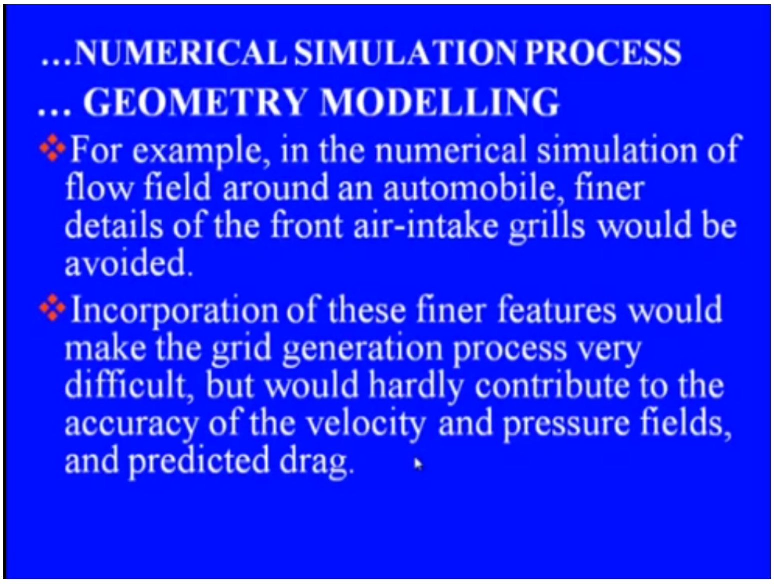


The first one is geometry modeling and since we using a computer to solve our problem we need a computer to presentation of our physical prototype or problem domain and for most of engineering problems please note that it may not be possible or even desirable to include all the geometric details of the system and its geometric model used for CFD analysis.

And I would like to emphasize here again that we have to prepare maybe an engineering design cycle or separate geometric model for CFD analysis compared to what we might use in stress analysis based on the aims of the simulation and the analyst has to make a careful choice regarding the level of intricate details of the geometric features, which are to be incorporated and which ones are to be left out because if we incorporate all the intricate details we have to be very careful during grid generation process.

And grid generation process might become unduly painful without much gain in the terms of the accuracy of the numerical solutions.

(Refer Slide Time: 06:24)

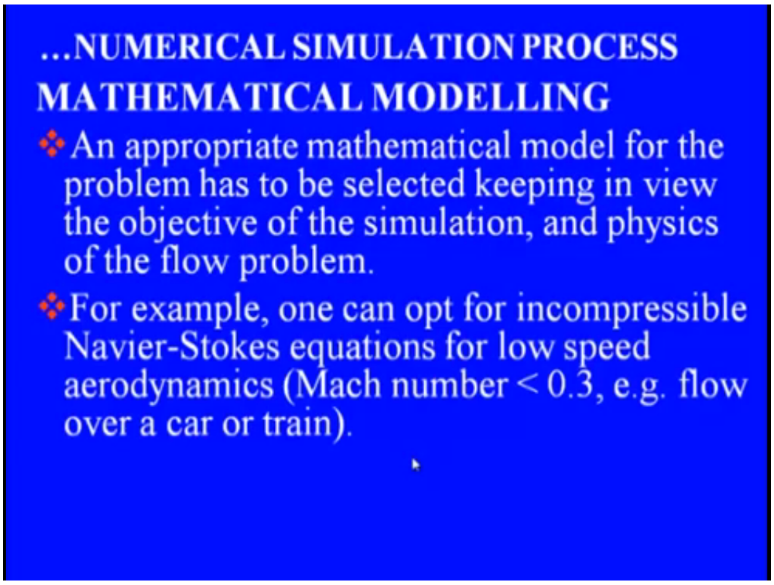


Let us have a look at a simple example of the numerical simulation for the flow field around automobile, suppose you want to design a new car and you want to optimize or reduce the drag on it. What are the features of the outer geometry which we have to incorporate in our geometric model, which would be used in CFD analysis? We are interested in finding out the flow field around the vehicle, but then the outer surface vehicle has to be modeled.

Now do we need to provide all the intricate details of the front air-intake grills or all the protrusions, which we have in our physical model, is that desirable? Usually not, we will simply ignore many of the finer geometric details, which are there in the prototype because incorporation of these finer features would make our grid generation process very difficult. We have to choose very fine grids close to those finer protrusions.

And that fine grid will only increase the cost of computation without contributing to the accuracy of the velocity and pressure field and the predicted drag, which is the aim of our simulation.

(Refer Slide Time: 07:45)

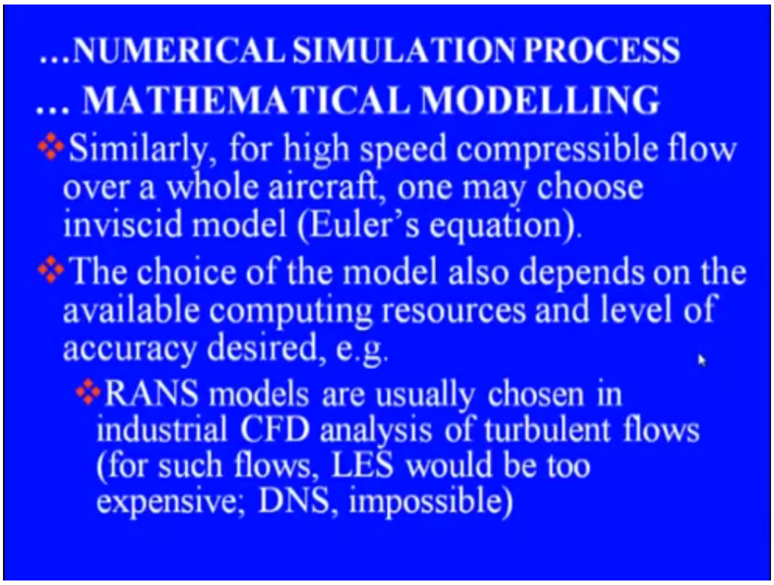


Next, let us have a look at the mathematical modeling step. We have to choose an appropriate mathematical model for the problem and the mathematical model will depend on the objective of the simulation and the physics of the flow problem. For instance, if we want to simulate the flow over an automobile or a lowest speed train hereby lowest speed train I mean the train is running at less than 400 kilometer per hour.

Since such situations what we see Mach number would be less than 0.3 and we can do away with or we would be happier if we just take what we call incompressible Navier-Stokes equations. We will have detailed look at the incompressible and compressible Navier-Stokes equations in our next module. So for lowest speed flows we do not have to consider the compressible effect.

We can solve a simpler model, which is incompressible Navier-Stokes equations.

(Refer Slide Time: 08:45)



On the other hand, if you are dealing with high speed compressible flows, say for instance we are dealing with hypersonic flows, in such situations for the initial design purposes we may simply ignore the viscous effects and we can go with an inviscid model in fact that what is routinely done in the aircraft industry that the initial simulations are based on the Euler's equation.

And sometimes simplified version of Euler's equation what we call full potential equations which are easier to solve compared to say the Euler's equation of full Navier-Stokes equations and the choice of the model will also depend on the available computing resources and the label of accuracy results for instance let us consider this (()) (09:28) turbulent flow problem.

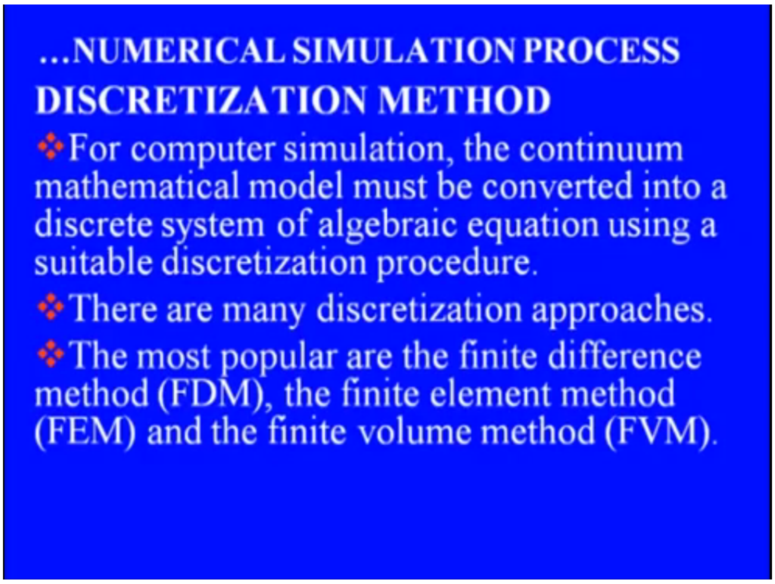
Suppose we are dealing with the flow through a turbine maybe want to model that flow for industrial design applications should we go for what we call Reynolds averaged Navier-Stokes simulations, which involve modeling of perturbations in the flow or should we go for more accurate larger simulations where modeling is used then only for what we call isotropic small scale areas or should we model all the scales of the flow, which is what is there in direct numerical simulation.

Now without going into further details we will discuss the details of RANS LES and DNS in the later modules just be aware here. The direct numerical simulation for industrial CFD analysis for turbulent flows are the same in turbine (()) (10:23) in rivers or channels is impossible with present stays of the development in computer science, computer hardware and software. So DNS is simply ruled out and the volume of the data, which we would get that would very, very difficult to handle as well.

So DNS is ruled out in first situations for design cycle what we would rather go for is we will go for less accurate, but freezable numerical simulation based on trans model. Large Eddy simulation are again too expensive and we used only maybe for the refinement of the final design, which would have been obtained from iterative simulations based on the RANS model.

So this way I have got availability of resources and they have simulations and the level of accuracy, which we desire that will also dictate the models which we have to opt for.

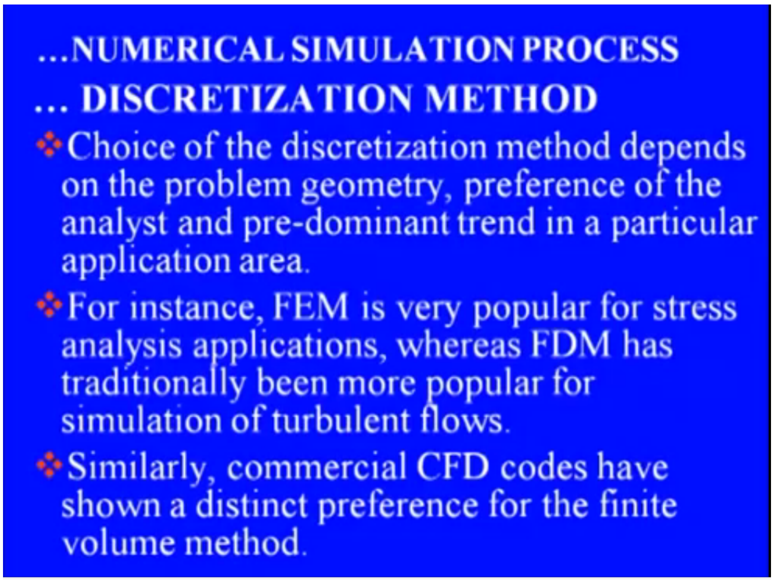
(Refer Slide Time: 11:30)



Next discretization method for computer simulation we have to convert our continuum mathematical model into discrete system of algebraic equations because that is what a computer can handle and we have to choose a suitable discretizing procedure and there many different procedures are available. In CFD the most popular ones are finite difference method, finite element method and the finite volume method.

We will have a detailed look at each of these methods in the later modules of the course. Today's lecture also will have briefly touch upon what these 3 methods mean? And how they are used?

(Refer Slide Time: 12:07)



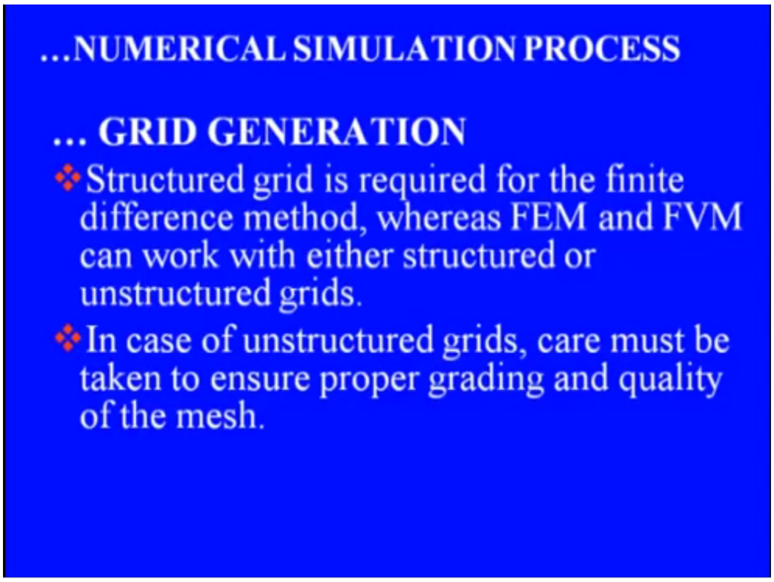
Which discretizing method would be chosen that would depend on the problem geometry. For instance, if we have got fairly simple Cartesian geometry, the easiest method to use would be finite difference method. For complex geometries, easiest one would be to go with an unstructured grid based finite element or finite volume methods so this one aspect geometry's one aspect.

Other is preference of the analyst that the analyst might prefer one particular method over other and this is also dictated by the predominant trend in a particular application area. For example, finite element method is very popular for stress analysis applications. Is not that we cannot use finite difference or finite volume method for stress analysis. Finite difference is the one which were used in early days of stress analysis by a physicist.

Similarly the finite difference method was used initially for modeling and simulation of turbulent flows so still finite difference method are traditionally very popular compared to finite element method in numerical simulation of turbulent flows. Commercial CFD course which have to deal with complicated geometrics and they have to satisfy a number of clients from different areas of application.

They have a distinct preference for finite volume method because this method provides the flexibility for the element in terms of handling complex geometries so this is why the preference has been for finite volume method in majority of the CFD course.

(Refer Slide Time: 13:51)

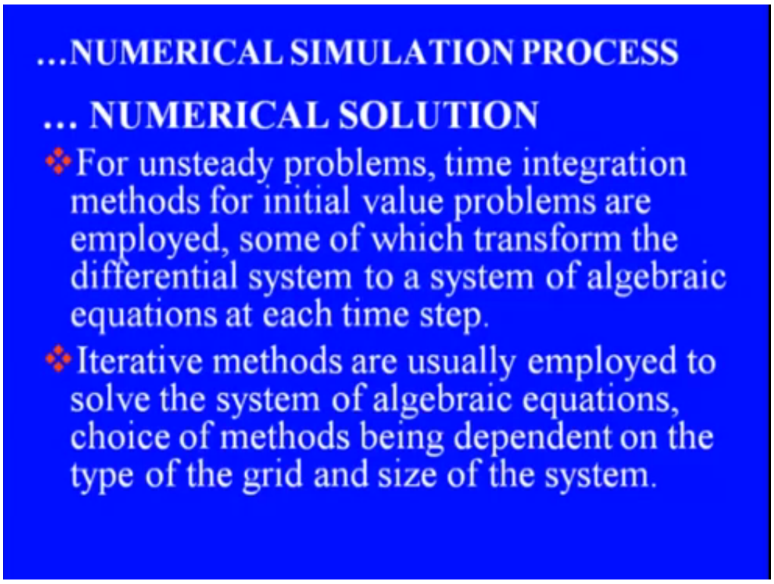


The next process which is the depending on the choice of discretizing procedure, we will have to generate a suitable grid to discretize your problem domain and the choice for the grid or mesh would depend on the discretizing method, which we have chosen. It will also depend on the problem geometry and the physics involved.

For instance, we are dealing with transonic flows or hypersonic flows over the wing bodies, our mesh are the grid so refine and have to resolve the shock waves, which are present away from the solid surfaces close to the body they will definitely have much finer grid but even we have to provide finer grids in those areas where shock waves are likely to appear. So the physical problem also dictates the design of the grid which we would use in our numerical simulation.

And whether we would use structured grid or unstructured grid that will also depend on the choice of discretizing scheme. For finite difference method, we have to go for unstructured grid. There is no other option available but with finite element and finite volume method, we can use either a structured grid or unstructured grid. In case if you have opted for finite element or finite volume methods based on unstructured grids, we must take proper care to ensure the proper grading and the quality of the mesh.

(Refer Slide Time: 15:22)



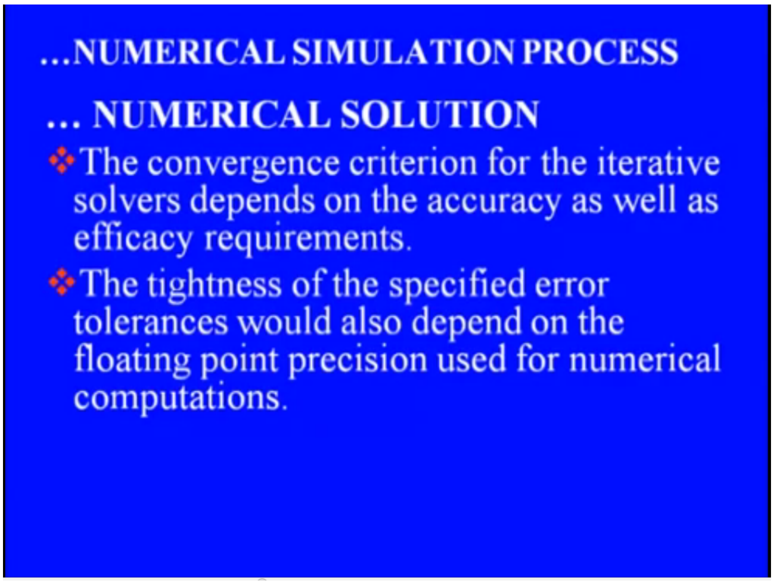
Next let us have a look at the numerical solution process. This deals with the solution of our discrete equations. We had applied our discretizing method chosen a suitable grid and on that grid when you apply the discretizing method, our continuum problem is transformed into a system of discrete equations, the other ones which are to be solved and the system discrete equations could be a system of ordinary differential equations in time if you are dealing with time dependent flows.

And these would be a system of algebraic equations for steady state problems. Now if you had time dependent problem, then we have to apply it in the approximation scheme what we call time integration schemes to convert this ordinary differential equations into a system of algebraic equations at each trimester. There are some time integration techniques wherein we not have to solve a system of algebraic equations.

That is to say if you using explicit time integration methods we might be able to find the solution of the problem using the straight forward formula, but with implicit methods we would again get a system of algebraic equations at each trimester. So solution of sparse algebraic system of equations that lies at the heart of computational mechanics or CFD simulations.

And usually the size of subsystems would be very large and is very difficult to apply what we call direct solution techniques so we will go for what we call iterative methods and the choice of the method would again depend on the type of the grid which we have used and the size of system.

(Refer Slide Time: 17:13)

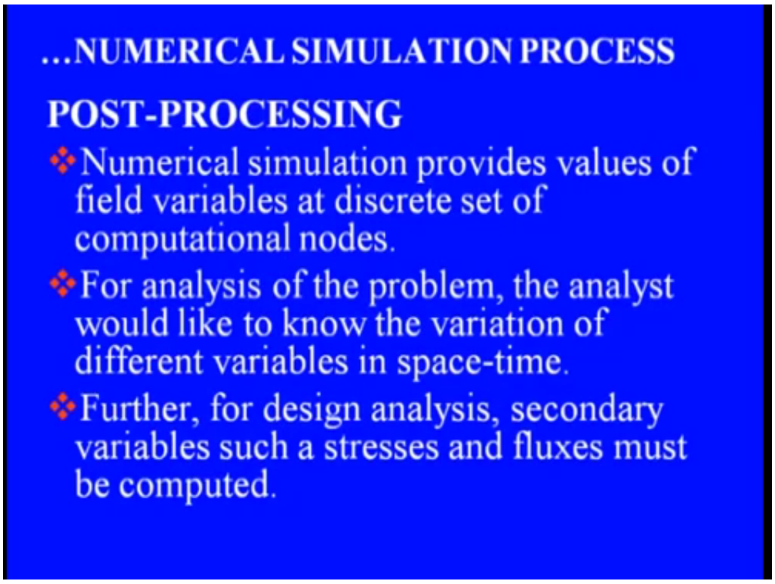


Now if you have chosen an iterative solver we have to set appropriate convergence criteria. Now this convergence criteria would depend on how much accuracy we require and how efficiently or how easily by investing how much computer time we want to obtain our solution. So that is what will govern the efficacy of the solution process and the tightness of the tolerance would also depend on the floating point precision used for numerical computations.

Say for instance you have chosen single precision, now in that case specifying a tolerance value of 10 to power-10 would be simply meaningless because our single precision floating point computations can provide us an accuracy of only up to 5 or 6 significant digits. So it does not make any sense to provide a tolerance titer then 10 to power-3 or 10 to power-4 at the most.

Wherein if you are dealing with double precision floating point computations, we can have a much tighter tolerance maybe of 10 to power-6 or 10 to power-8 for the iterative solution process.

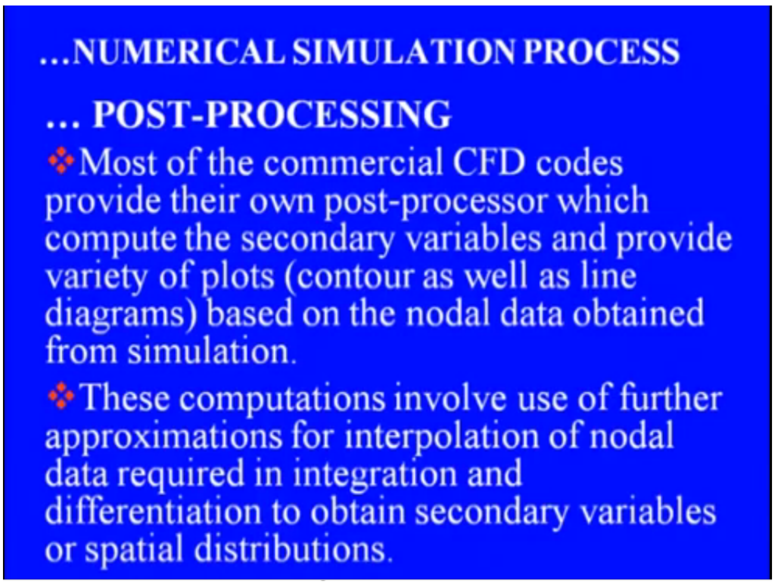
(Refer Slide Time: 18:22)



The last step in numerical simulation is what we call post-processing. The numerical simulation that gives us the values of flow variables at discrete set of computational nodes and it is very difficult to make out anything from looking at the numerical values at so many points. So what we would like to have I can analyst of the problem we would like to say the variation of different flow variables with the space and with time.

And for this purpose we need to perform what we call the post-processing of the data which has been generated by the CFD simulation. Similarly for design analysis, we might be interested in let say drag. We might be interested in stresses and heat fluxes. Now these are what we call secondary variables. These must be computed by strong the available flow simulation data.

(Refer Slide Time: 19:20)

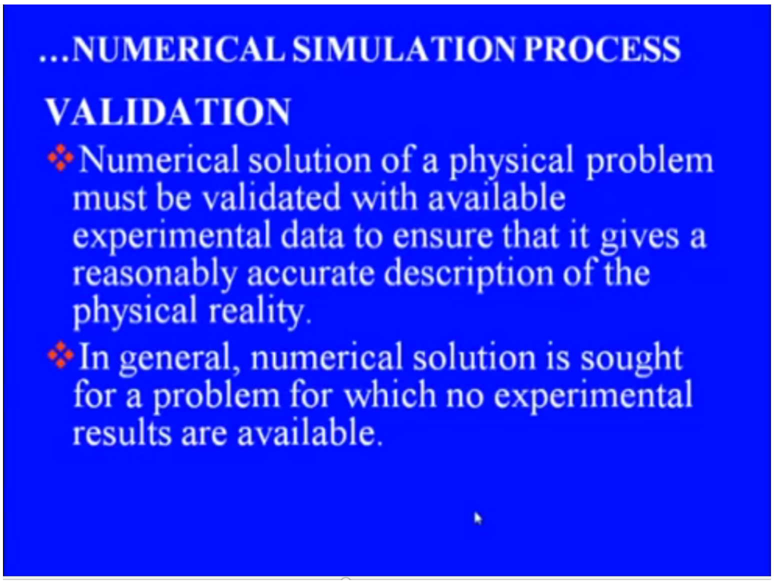


Now most of the commercial CFD course they provide their own post-processors, which compute the secondary variables and they will provide you a variety of beautiful plots, contour plots, line plots and so on which are based on the nodal data obtained from the simulation. Similarly suppose you have written your own CFD code, we did not have a post processor.

There are commercial post-processor or open source software available. You can just tailor your output to the input requirements of those software and obtain a similar type of post-processed data as let say Fluent or StarCD would provide you. Now please remember that these computations at the post-processing step they also involve further approximations because what we have from the CFD simulation is data at certain set of computation nodes.

And if you want to have the variation of the entire domain, we have to perform certain interpolations. We might have to perform differentiation flow variables to obtain fluxes and stresses. So there would be additional approximations involved in the interpolation of the nodal data for integration as well as differentiation to obtain the secondary variables and to obtain spatial distributions for our contour plots.

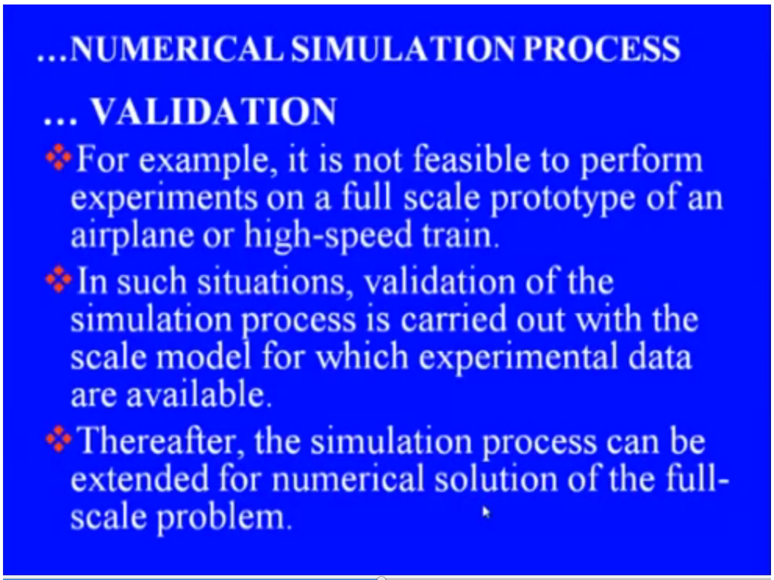
(Refer Slide Time: 20:40)



The last step is what we call validation. We also had a brief discussion about validation in the previous lecture and this is one of the most important step say that numerical simulation had given us a solution but does it accurately reflect the physical reality that we do not know unless and until we have validated it with available experimental data, but there is a catch here that in general we would be performing numerical simulation for a problem for which no experimental data is available.

It might be in altogether new design, which you have just come up with and you want to refine it further so there is absolutely no possibility of having an experimental data. Experiments will be performed on the prototype, which would be based on what you get after your design analysis is complete and you have come with reasonably final representation of the design of your product.

(Refer Slide Time: 21:42)

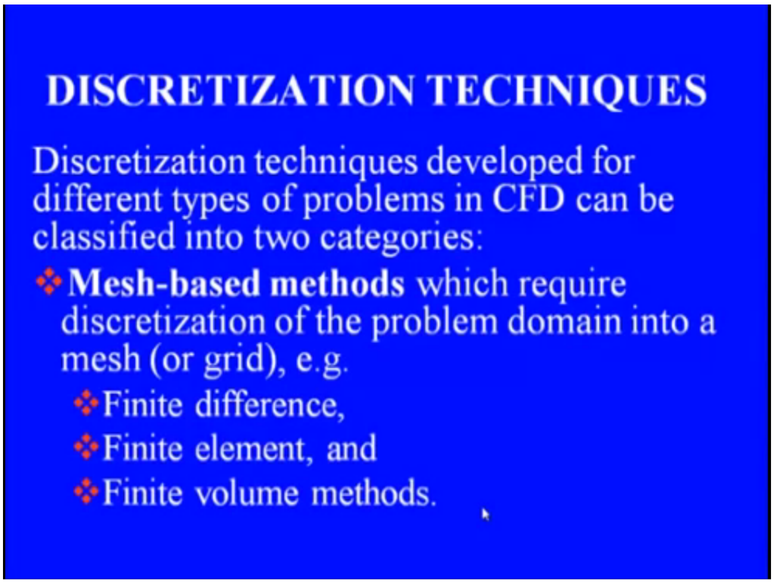


So now what do we do in such situations. So in such situations what we would do is we will validate the problem based on the experimental data for a similar system or subsystem. For instance, we have designed an air plane or high speed train now for this one way to validate would be come up with this scaled model, perform a numerical experiments or numerical simulation on scaled model.

Try to perform the experiments in wind tunnel for the scaled prototype and use that experimental data to validate your numerical simulation on this scaled model and then go for numerical simulation on full prototype. So this is one way that is used normally the most practical way which is opted in industry for the validation of numerical simulation data and once you have validated our results, we can perform the simulation confidently for the full scale problem and optimize our design.

Next let us have a brief look at the discretizing techniques, which are used in CFD and computational mechanics.

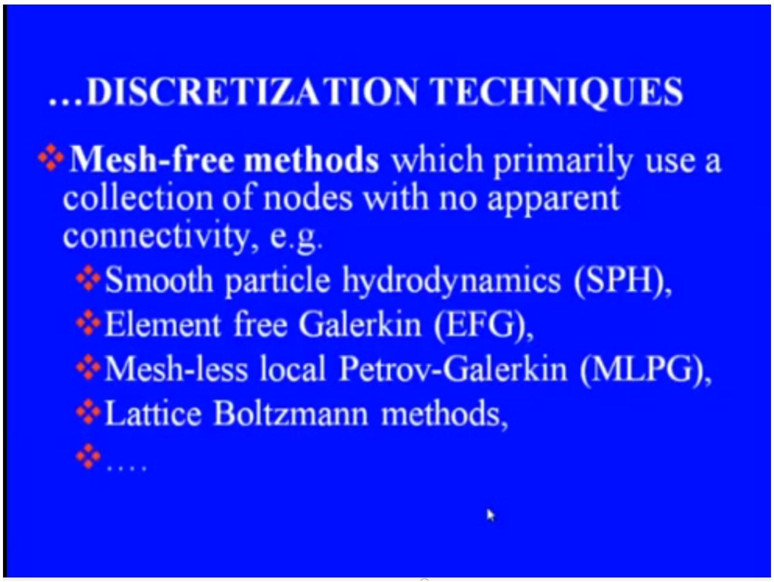
(Refer Slide Time: 22:53)



And we can broadly classify these techniques in 2 categories. The first one what we call as mesh-based methods, which require discretizing of a problem domain into a mesh or grid and the popular examples of finite differences, finite element and finite volume method is also few of the methods notably bound element method as well which require a mesh. By a mesh what we mean we will have a set of discrete points but though it is skit points would be interconnected.

We also need the connectivity information in application of these methods. The points are not arbitrarily distributed in the domain so those interconnections that leads to what we call finite elements or finite volume cells or gird elements.

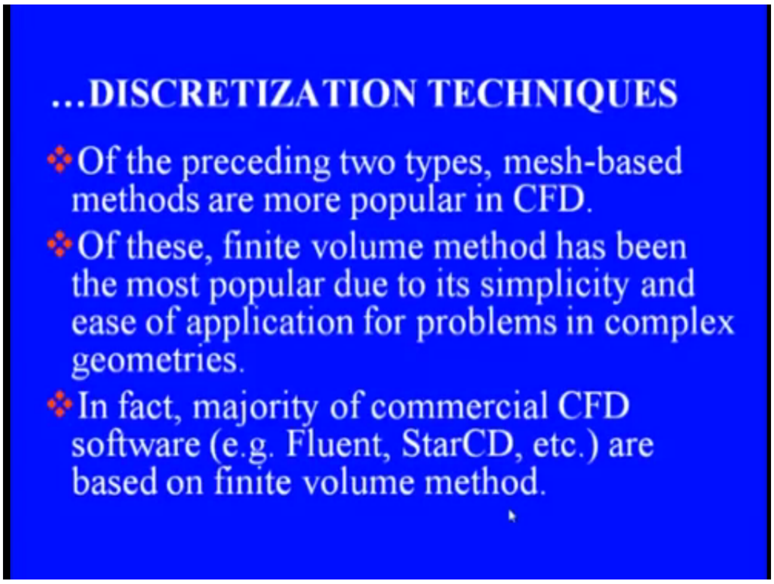
(Refer Slide Time: 23:44)



In contrast to a mesh-based method, we have also got a category of method which we call mesh-free methods which primarily use a collection of nodes with no apparent connectivity. The computational nodes could be arbitrarily distributed in the solution domain. The connectivity information is not required in the formulation or in discretizing process.

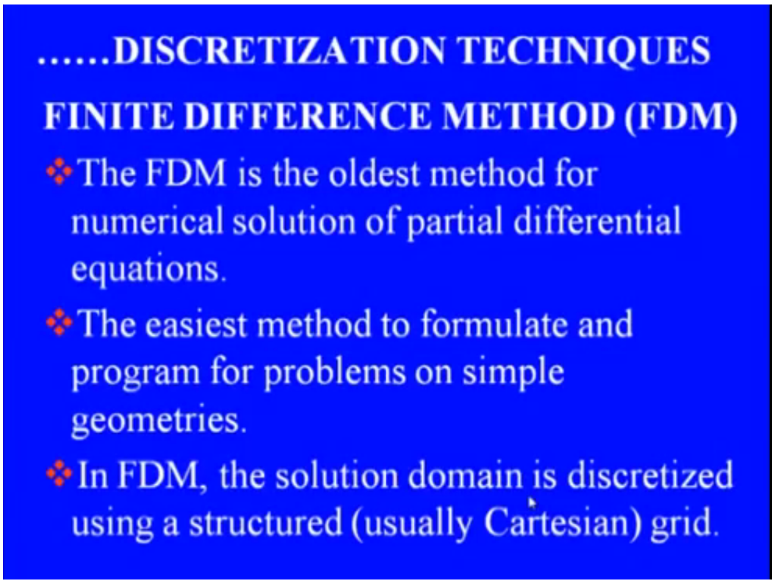
And some of the popular methods, which have been discussed are smooth particle hydrodynamics, element free Galerkin method, Mesh-less local Petrov-Galerkin method, Lattice Boltzmann method and there are many, many such schemes. If you are interested you can have just have a look in the literature. You can simply type Google search of mesh-free methods and you will see there are 100s of variants of different mesh-free methods available in literature.

(Refer Slide Time: 24:38)



But when it comes for CFD, mesh-based methods are the most popular and of these finite volume methods have caught the attention of the commercial developers because its simplicity and easy application for problems and complex geometries so that is why majority of commercial CFD software for instance ANSYS Fluent or StarCD they are all based on finite volume method.

(Refer Slide Time: 25:04)

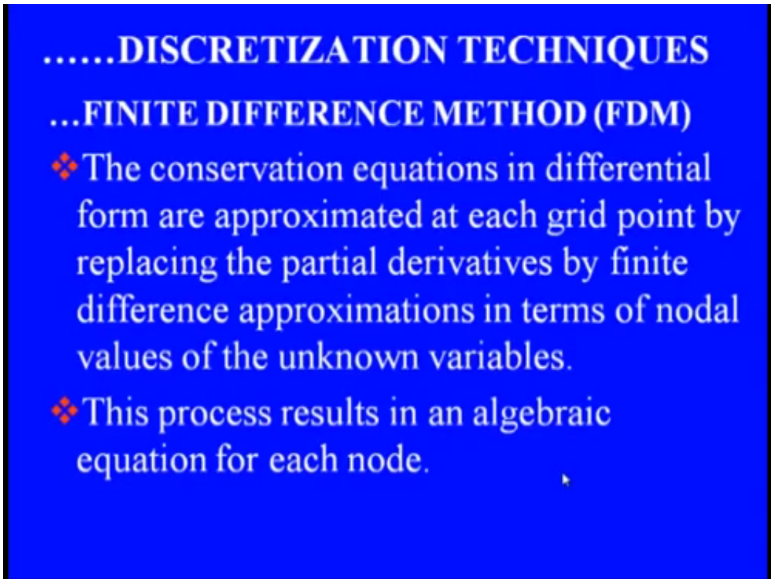


Now let us have a brief look at the 3 mesh-based methods for instance finite difference, finite element and finite volume method in bit more detail. We will take these methods formally in each we will dedicate one module each to finite difference, finite volume and finite element method later on in this course. Now finite difference method is the oldest method for numerical solution of partial differential equations.

It dates back to 1800. It is also the easiest method to formulate for the problems on simple geometries though with recent developments in immersed boundary method, have made this method equally capable to solve problems on arbitrary complex geometries or grid still remains because simple Cartesian grid, but there are techniques available using which we can module our curved geometries to the specified tolerance or accuracy.

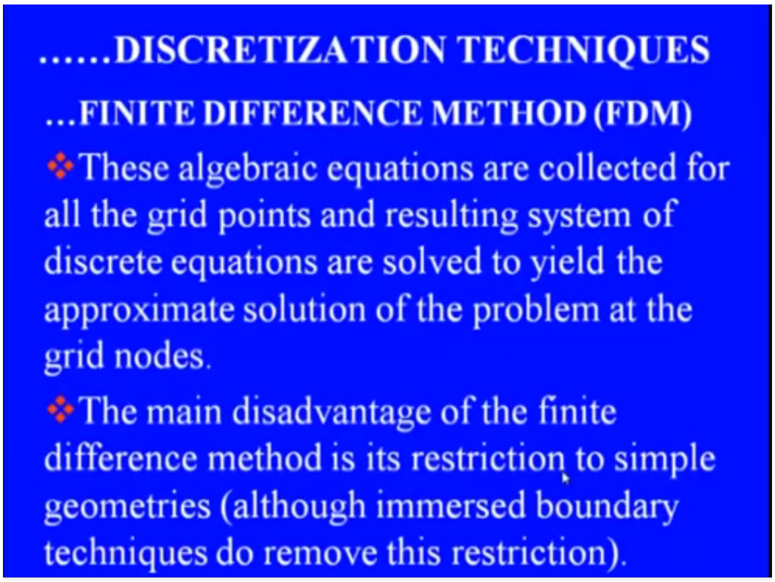
Now what do we do in finite differences that are solution domain is discretized using a structured grid usually a Cartesian grid in some instances we can go for a body fitter grid wherein finite difference discretizing is applied to the transformed equations.

(Refer Slide Time: 26:25)



And we use the conservation equation in differential form and what we sensibly do is at each grid point in our computational domain, replace the partial derivatives why their finite difference approximations which would be define in terms of the nodal variables at the neighboring points. So this particular step transforms our partial differential equation into an algebraic equation at each computational node.

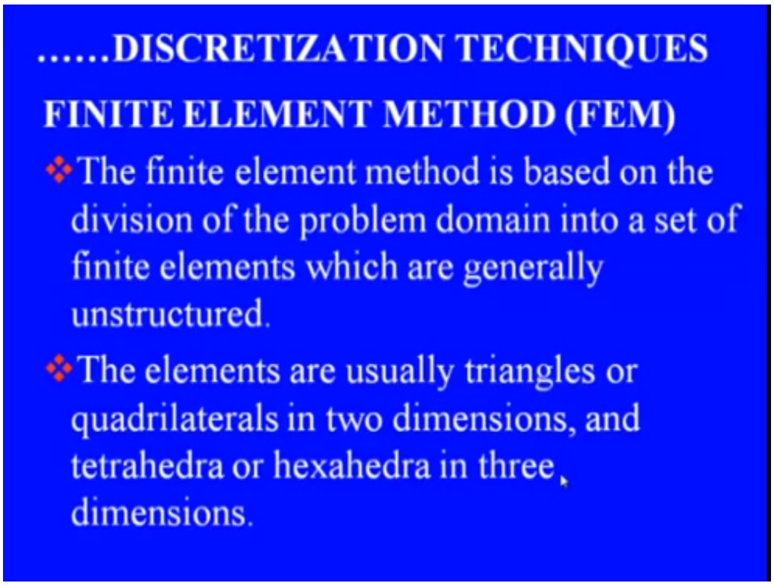
(Refer Slide Time: 27:00)



And then we can collect all such algebraic equations for all the grid points and then the resulting system discrete equations are solved to yield approximate solution of the problem at all the grid nodes and once we know the solution to all the grid nodes we can use the post-processing technique to obtain the secondary variables and the distribution throughout the domain.

The main disadvantage which used to be earlier before the advent of our wider popularity of immersed boundary techniques was that finite difference method are restricted to simple geometries. Please remember this immersed boundary techniques, they have removed this restriction and we are no more limited by the complexity of the geometries while if you want to use finite difference method as our preferred discretizing technique.

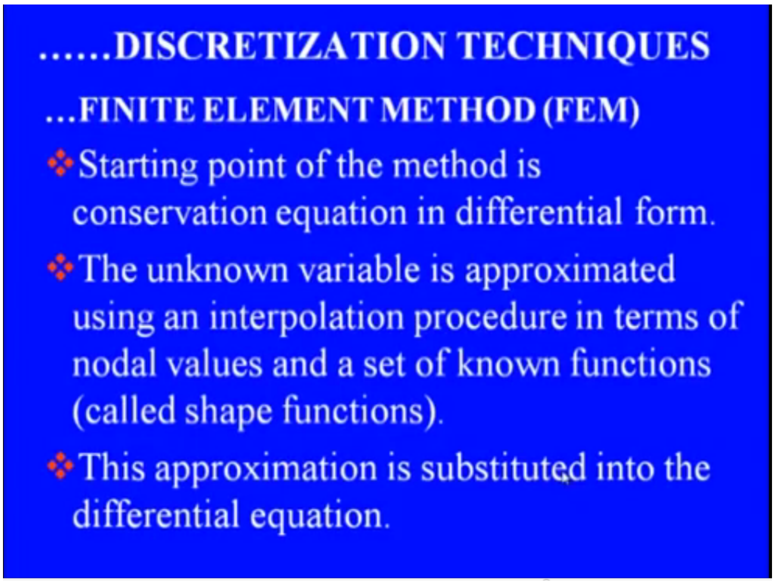
(Refer Slide Time: 27:53)



Next just have a look at the beautiful method called finite element method. This is the method which has been credited with the revolution in computational mechanics as well as in computation fluid dynamics and what we do here in finite element method is to divide our problem domain into a set of finite elements, they may be structured or they might be unstructured.

And the elements usually triangles or quadrilaterals in 2 dimensions or tetrahedra or hexahedra in 3 dimensions. In fact, in the beginning triangular elements were the most popular because using them is very easy to generate grids on arbitrarily complex geometries and that was for the reason why majority of the grid generation scheme, which were developed for finite element they were referred to as triangulation techniques.

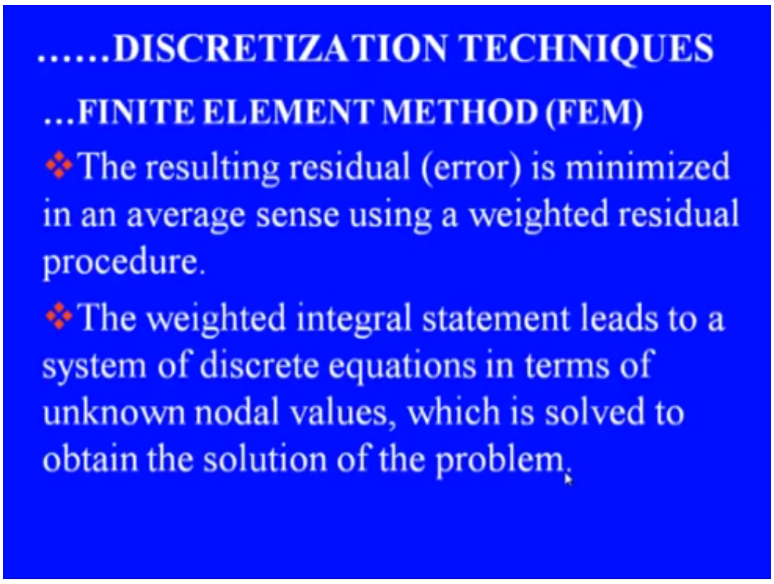
(Refer Slide Time: 28:43)



And the starting point of the method is conservation equation in differential form from there we transform it into an integral form and thereafter obtain a system of discrete equations. The most popular process here is that we first approximate the unknown variable using an interpolation procedure wherein the nodal values are approximated using a set of known functions.

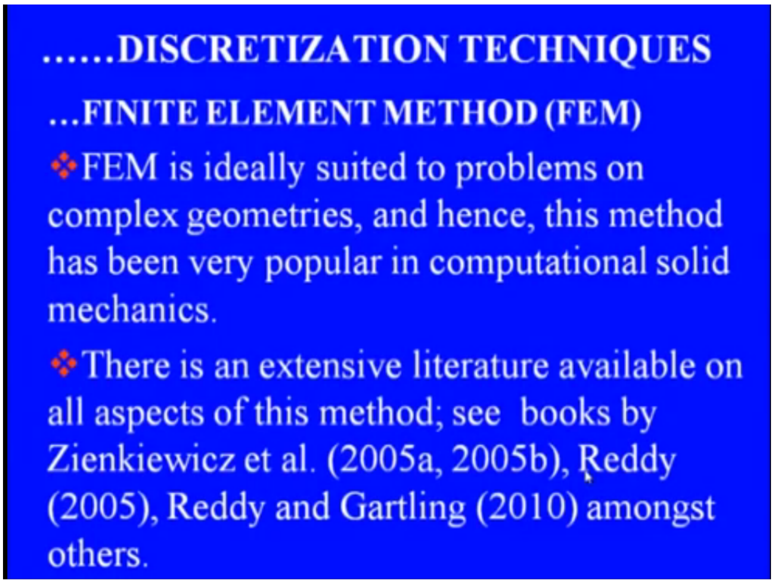
Rather the problem variable over the domain or over in particular element that is approximated in terms of unknown nodal values and a set of known functions. These set of known functions are called interpolation functions, shape functions or trial functions. Now this approximation is substituted on a differential equation, which will not to be satisfied exactly and will lead to an error.

(Refer Slide Time: 29:50)



Now this resulting error is termed as residual and we want to minimize this residual in an average sense using a better residual procedure and this particular process of trying to minimize their residual in an average sense using a weighted integral statement leads to a system discrete equation in terms of unknown nodal values. So we have already got our discrete system, solve it to obtain the solution at the nodes.

(Refer Slide Time: 30:17)

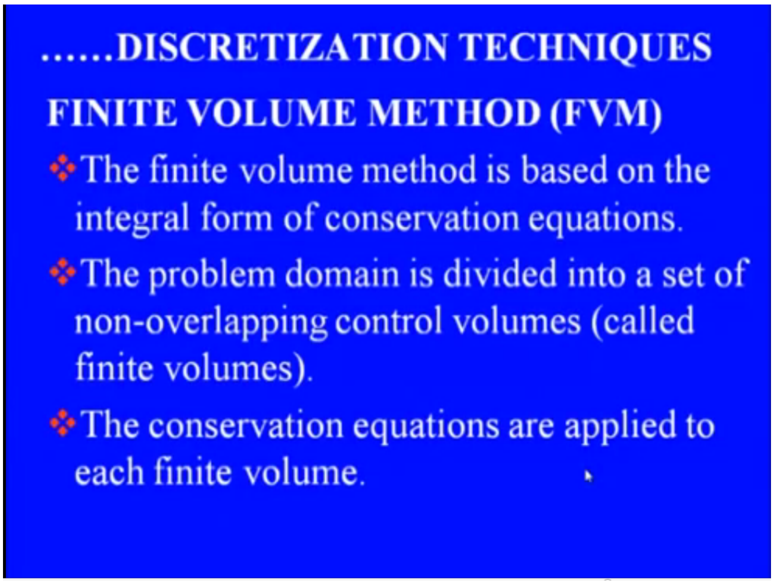


Now once the values of nodes are available, we know the interpolation functions, using them the solution can be obtained at any point in the problem domain. The shape functions can also be used to obtain secondary variables whether they require differentiation or integration it really does not matter. So this finite element method is ideally suited to problems on complex geometrics.

And hence this method was very popular still very popular in computational solved mechanics. In fact, 99.9% codes for computational solved mechanics are based on finite element method. There is extensive literature available on all the aspects of this method. I would just refer to you few books like you can refer to the classics by Zienkiewicz et al or you can refer to the book by Reddy.

There are many, many books available on finite element method. In this particular course, we will just briefly touch upon the finite element method in one of the modules. For details you can always refer to these books.

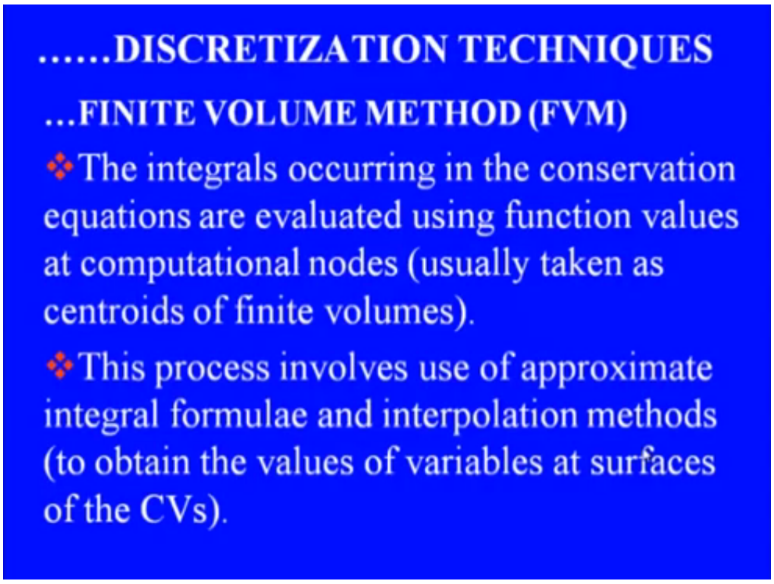
(Refer Slide Time: 31:16)



Next let us come to the finite volume method which is one of the most popular methods used in commercial CFD applications and this method is based on the integral form of conservation equations not differential form and herein we follow a procedure fairly similar to what is used in finite elements that is divide the problem domain into set of non-overlapping control volumes.

So integral form of conservation equations are defined over an arbitrarily complex problem domain. So same integral equations also hold good for they will hold in each of these finite volumes so that is the basis of this finite volume method.

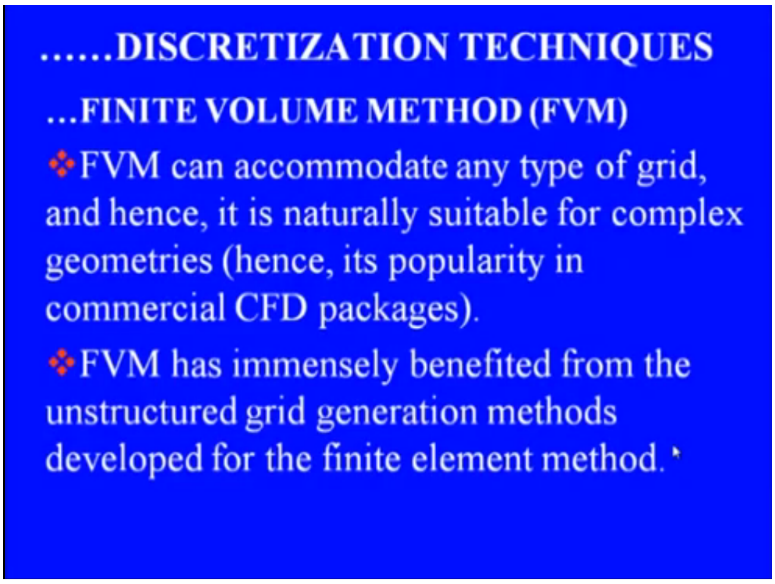
(Refer Slide Time: 32:11)



So we can apply our conservation equation over each of these finite volumes and all that we do is the integers which would occur in the conservation equations, they are evaluated using function values at computational nodes which are yet unknown, these values are unknown but in terms of these unknown values we will approximate the integrals with their volume integrals or surface integrals usually computational nodes has taken as the centroids and this process of approximation involves certain interpolation formulae and integral formulae to obtain the value of variables at the surfaces of the CVs.

We will look at the details of this process when we come to finite volume method in later modules of the course.

(Refer Slide Time: 32:50)



Now finite volume method can accommodate any type of grid and hence it is naturally suited for complex geometries and it has immensely benefitted from developments in finite element method specifically developments on structured grid generation techniques.

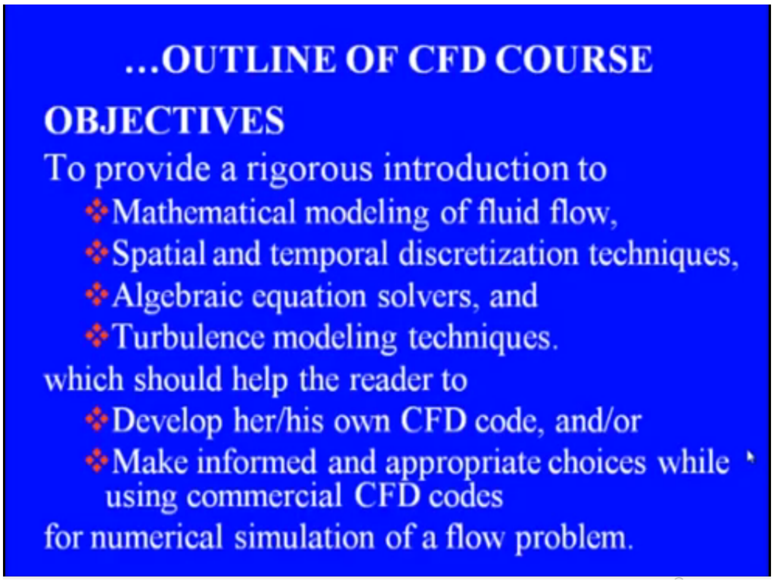
(Refer Slide Time: 33:07)

Now let us have a look at the outline of the course what we are going to cover in this particular introductory course. Now remember computational fluid dynamics has become an essential tool in analysis and design of thermal and fluid flow problems. So whether you are going to take up a carrier in industry or in academy, you would use these tools in design as well as in research work.

Now even if you use in a commercial CFD code the correct use of CFD requires a thorough understanding of the underlying physics, you must know your fluid dynamics, you must know what are the things involved in mathematical modeling and you should also be aware at least and you should know the numerical techniques, which are involved in numerical simulation process.

As a user you must also be fully aware of the properties and limitations of the numerical techniques which are being employed in CFD analysis and this particular course has been designed to fulfill these objectives. So we will familiarize you thoroughly with mathematical modeling, physical problem and you would also become aware of the properties and limitations of each numerical technique which we use in our numerical simulation process.

(Refer Slide Time: 34:26)

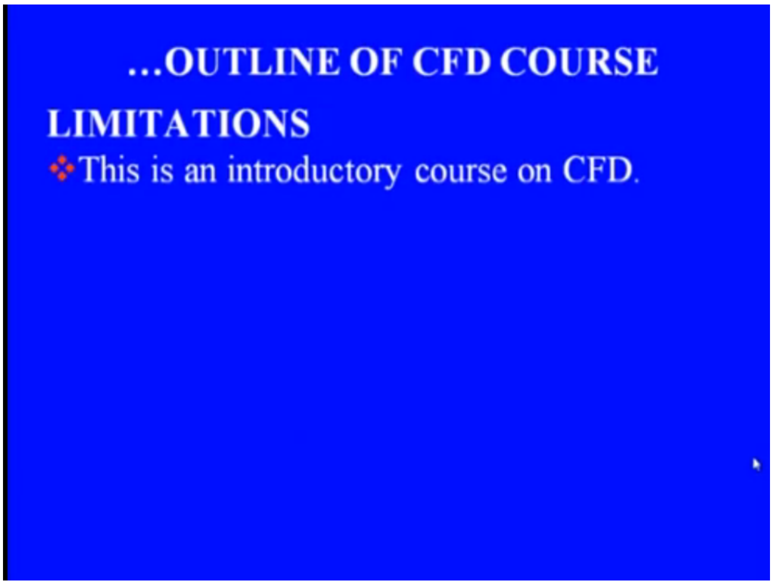


So to summarize this is the objective to provide a rigorous introduction to mathematical modeling of the fluid flow so that you become comfortable with the physics, spatial and temporal discretizing techniques for numerical simulation, algebraic equation solvers and turbulence modeling techniques, which are required for modeling and simulation of turbulent flows which are ubiquitous in practical applications.

And this rigorous introduction should help the reader to develop her or his own CFD code and or make informed an appropriate choices while using a commercial CFD code for numerical simulation of the flow problem. Otherwise, it will be worthwhile to rephrase a code, which was acquainted in finite element analysis that commercial finite element course they make a good engineer out of standing and a bad engineer outright dangerous.

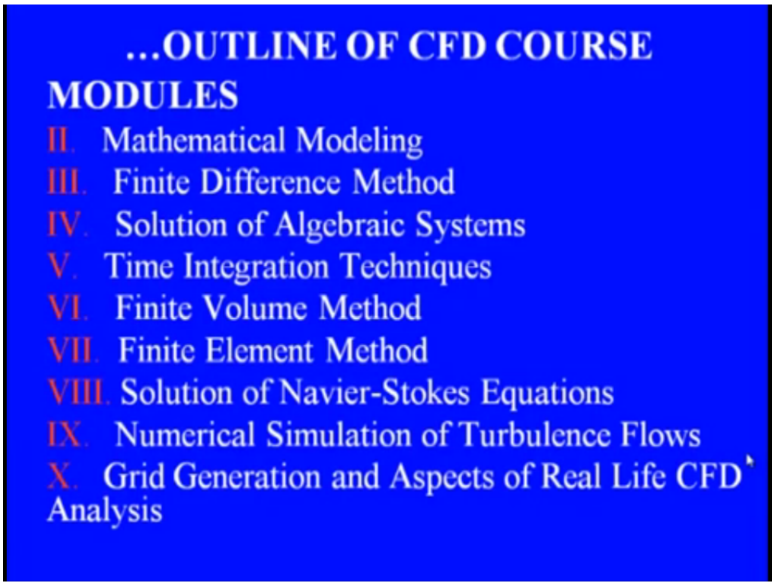
The same thing applies to the CFD codes as well so unless the analyst is thoroughly familiar with the physics and the mathematics which is involved in this numerical simulation they might not make correct or appropriate use of the CFD techniques, which are very powerful so these powerful tools must be applied with full understanding and that is the primary aim of this course.

(Refer Slide Time: 35:56)



Limitations, please remember this is an introductory course so we may not be able to provide a detailed coverage of each discretizing scheme or each topic but nevertheless we would refer you to the appropriate references, which will help you out.

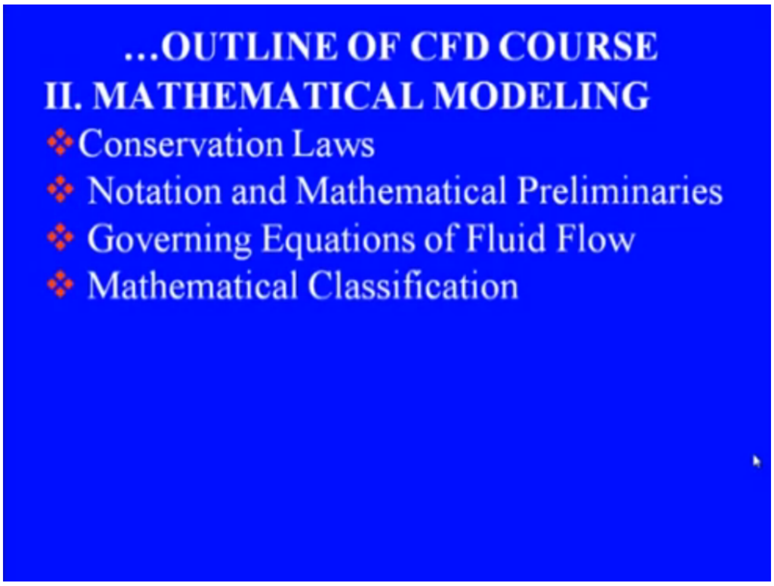
(Refer Slide Time: 36:09)



Now let us have a brief look at the modules of the course description. We will first start in the next module will take up mathematical modeling. Whatever mathematics will require for this course will provide a brief overview of that of the mathematical notation, few mathematical theorems and the conservation principles and how do we use those conservation principles to obtain the mathematical model for a flow problem.

So we are going to discuss that in detail. Then we will have a look at finite difference method in module 3, solution of algebraic equation in module 4, time integration techniques in fifth module, then we will proceed to finite volume method, finite element method then to Navier-Stokes equations and numerical simulation of turbulent flows and end we would briefly discuss the grid generation and the aspects of real life CFD analysis.

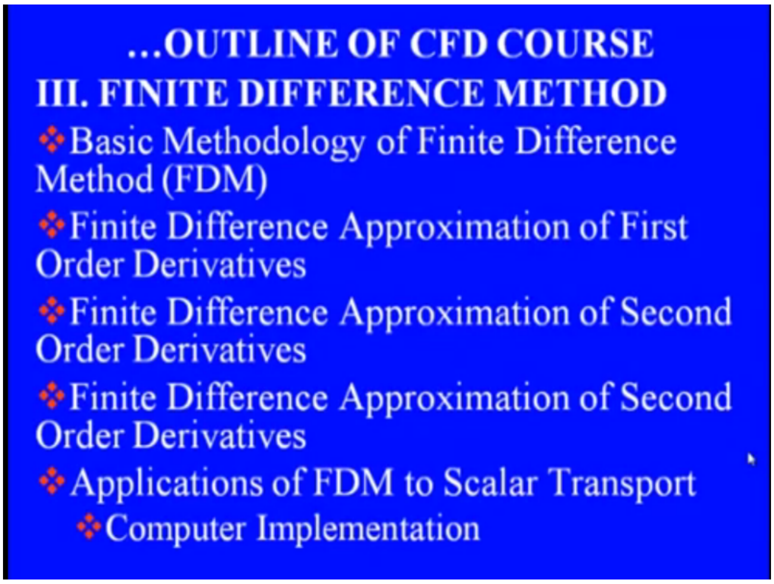
(Refer Slide Time: 37:09)



Now let us have a bit more detailed look at each of these modules. In mathematical modeling, we will discuss conservation laws. We will also discuss the notation which is commonly used in CFD literature and then we will take up governing equations of fluid flow starting from mass conservation, momentum and energy equation and obtain the corresponding integral as well as differential form of the physical laws.

Then we will also come up with mathematical classifications. We will look at what are boundary conditions that we need to module our flow problem.

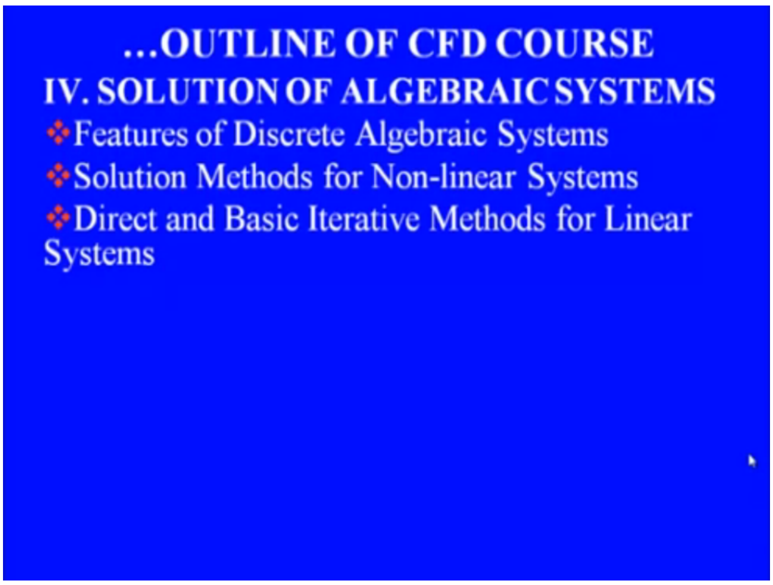
(Refer Slide Time: 37:39)



Then we will take the simplest discretizing technique or finite differences and we will have a detailed look at this method. What is the basic methodology? How do we approximate first order derivatives, second order derivatives, and the applications of finite difference method to scalar transport problem and we will also look at the computer implementation aspects of finite differences.

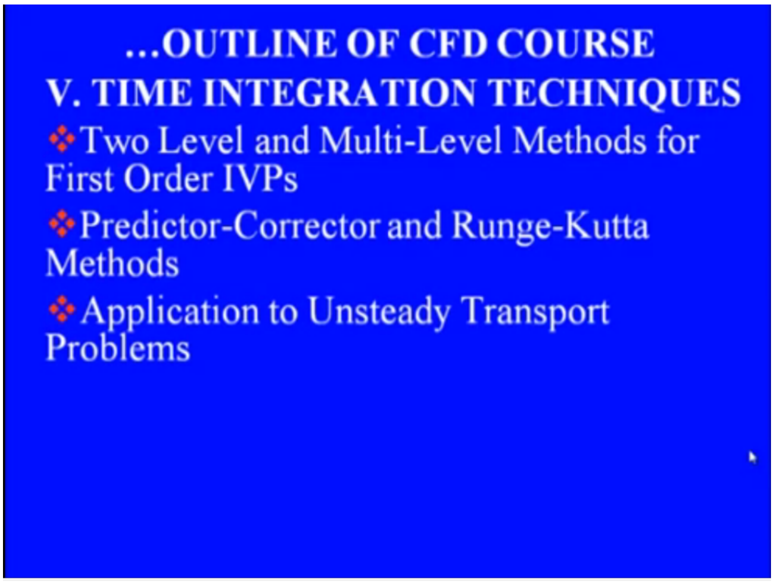
Now this computer implementation aspect would be detailed enough for you to help you to apply or write a code based not only on finite difference but also on finite volume and finite element method later on. Once we have covered one discretizing scheme, we have obtained our discrete system.

(Refer Slide Time: 38:30)



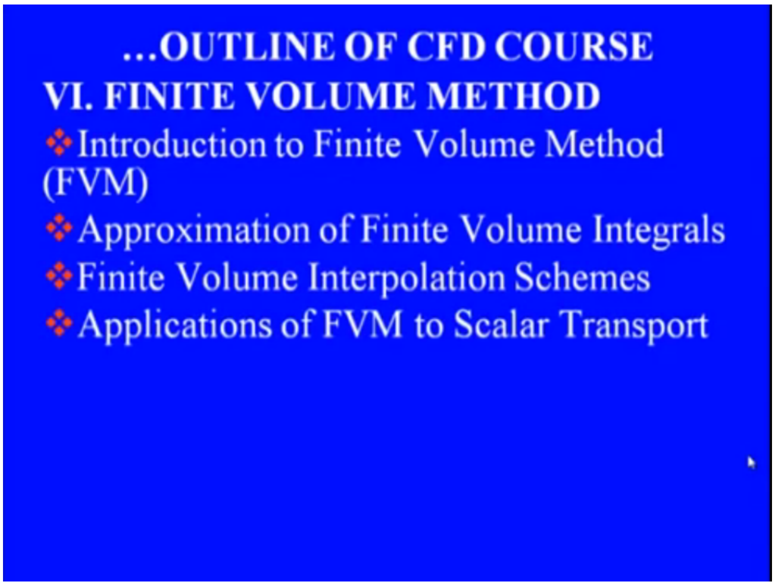
If it is in algebraic system, we would like to solve it so the 4th module we will discuss on the schemes, solution methods for nonlinear systems, then we will discuss direct and basic iterative methods for linear systems, which are used in CFD. We will also look at the family of methods which we call accelerated iterative methods and they are the ones which are most commonly used in practical CFD analysis.

(Refer Slide Time: 38:55)



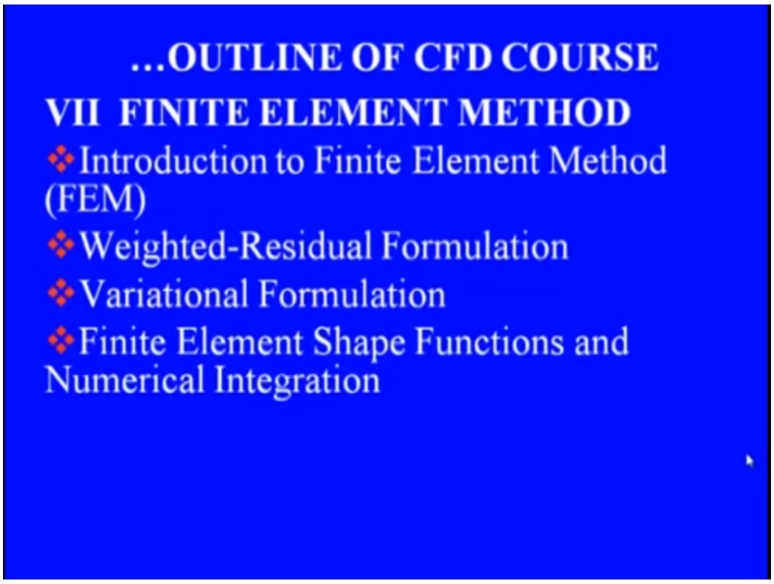
Next we will have a brief look at few popular time integration scheme for unsteady problems. You will have a look at 2 level or multilevel methods of first order initial value problems. We will look at predictor-corrector and Runge-Kutta methods and we will discuss the applications of these methods to unsteady transport problems and thereafter we will move on to finite volume method.

(Refer Slide Time: 39:10)



How do we approximate the integrals involved in finite volume technique? What are interpolation schemes involved? And then we would discuss how do we apply the finite volume technique to scalar transport problems?

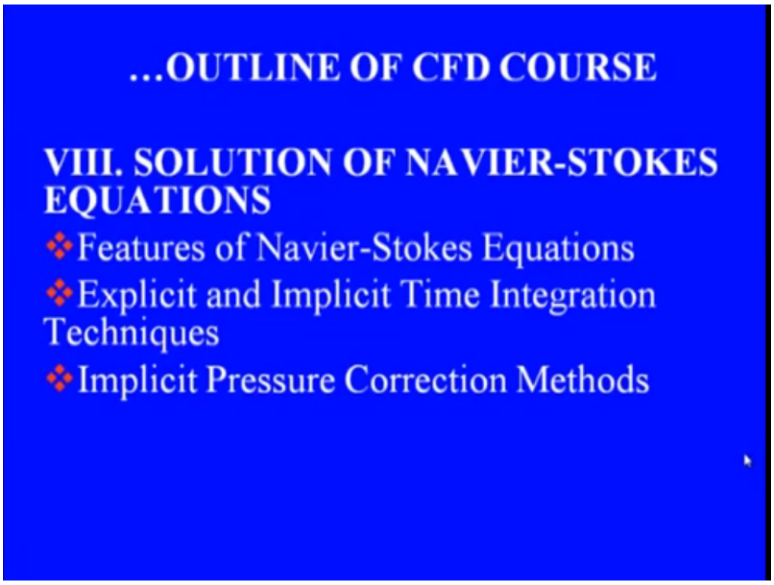
(Refer Slide Time: 39:27)



Then we would move on to finite element method a very brief introduction to finite element. We will have a look at 2 sets of formulation that is weighted-residual formulation, which is the most popular one and a variational formulation of the method. Then we will also discuss briefly the finite element shape functions and numerical integration for both 2 dimensions and 3 dimensional applications.

And then we would apply finite element method to scalar transport problem so these discussions would enable you to make use of finite element method to solve time dependent heat conduction equations and extended further for solution of flow problems.

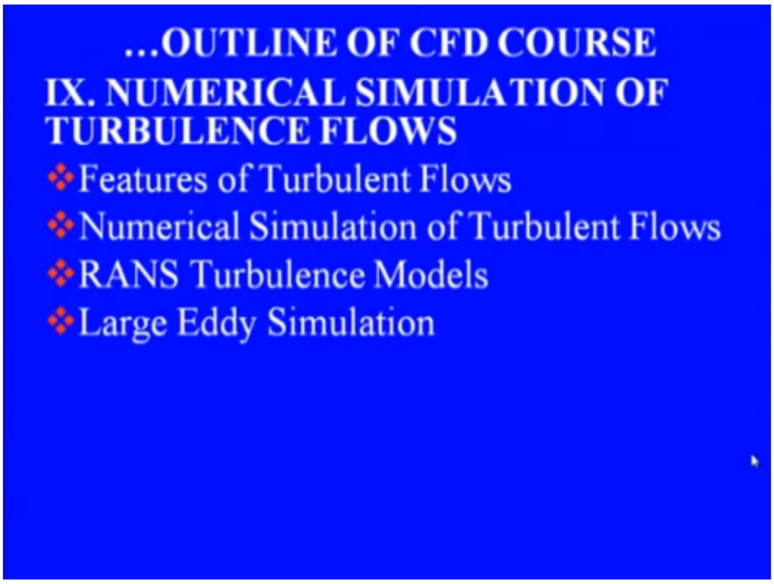
(Refer Slide Time: 40:04)



Once we have learnt our discretizing techniques, techniques for the solution of algebraic systems and techniques for solving time dependent problem in our previous modules then you would attempt this is the heart of the CFD we want to solve Navier-Stokes equations for this fluid flow. So how do we apply these techniques for solution of Navier-Stokes equation that forms the objective of module 8.

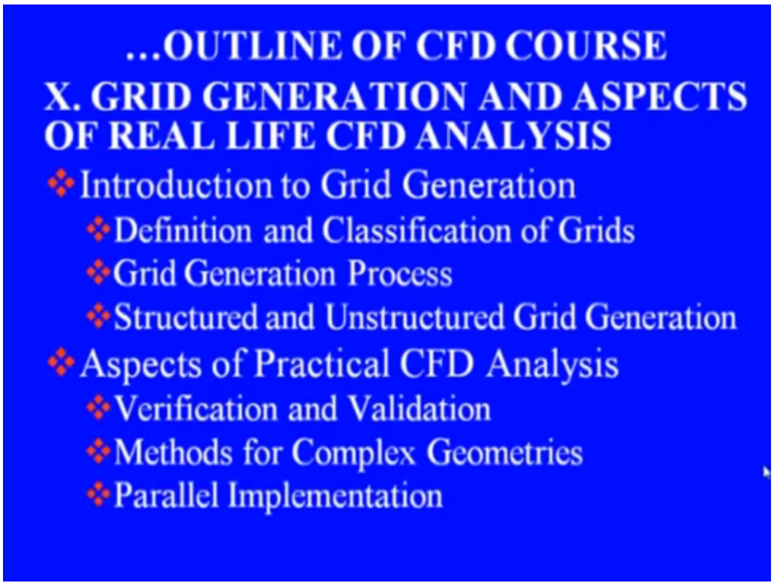
We will discuss the features of Navier-Stokes equation which differentiates from scalar equations and we will discuss both explicit and implicit time integration techniques. We will discuss the popular implicit pressure corrections methods and fractional step methods.

(Refer Slide Time: 40:50)



We will then move on to the numerical simulation of turbulent flows and we will very briefly discuss what are basic features of turbulence? and what are numerical simulation techniques which are used for turbulent flows? In particular, we will have look at the RANS turbulence model. We will also touch upon the large Eddy simulation.

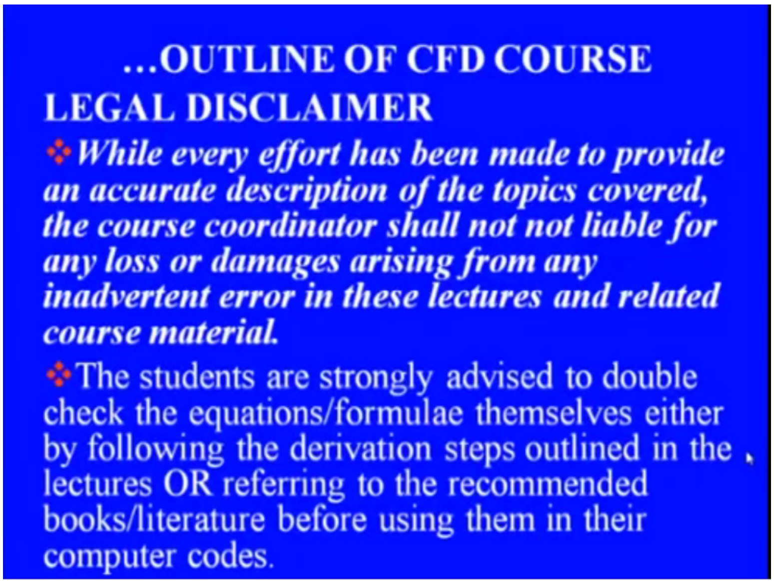
(Refer Slide Time: 41:11)



And in the last module, we will briefly touch upon the grid generation. We will discuss the definition and classification of grids, grid generation process both structured and unstructured grid generation and thereafter will have a look at few very important practical aspect which you must keep in mind as a CFD analyst that are related to verification and validation process.

And which are the methods will you use for complex geometries and most importantly in the end we will cover or discuss the parallel implementation, which is not being routinely used in CFD analysis.

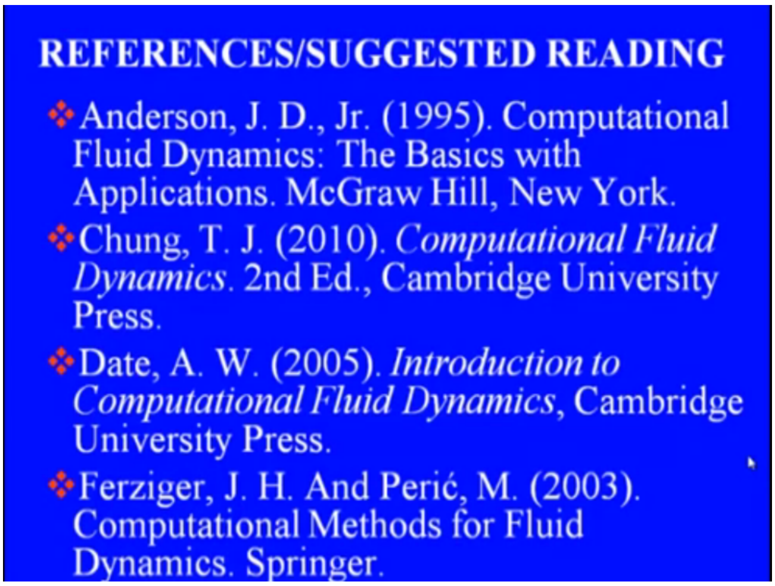
(Refer Slide Time: 41:47)



Now I would like to point out few things first legal disclaimer. Though we have made every effort to provide an accurate description of all the topics, the course coordinator shall not be liable for any loss or damages arising from any inadvertent error in these lectures and related course material.

So the students are strongly advised to double check the equations of formulae themselves either by following the derivation steps outlined in the lectures or referring to the recommended books or literature before using these formulae in their own computer course. Now few references of suggested books that are my preferences that are many very good books available on computational fluid dynamics.

(Refer Slide Time: 42:45)

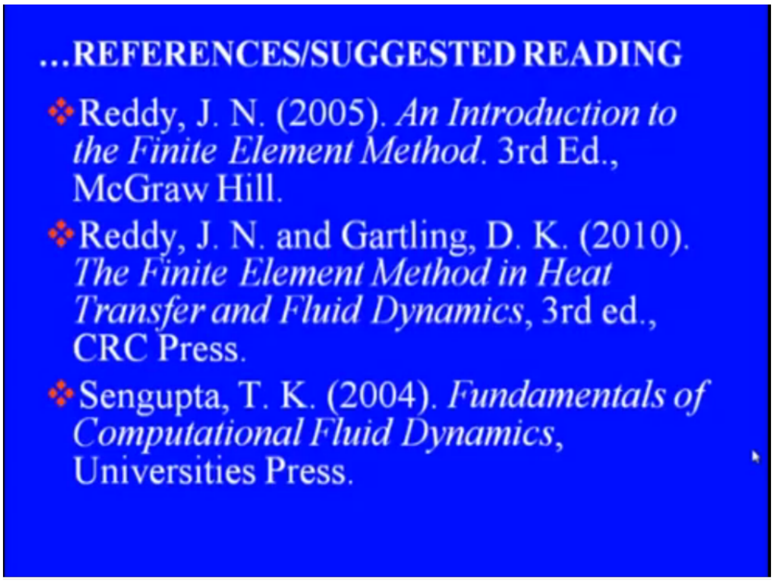


Here I provide a list of the ones which I have referred which I like the first one is Anderson J.D. this listing is not in the order of any priority, it is alphabetical listing from the author name. Anderson's book of computational fluid dynamics the basics with applications. This one is the most fascinating introductory books on finite difference method in this application in flow problems, the tilt test towards the aerospace applications.

Then book by Chung on computational fluid dynamics. It is in fact a compendium I would say one of the most comprehensive books available out of CFD. It is not introductory text, Anderson's book is an introductory text, but the book by Chung can help you out by any technique, which we discuss or which had been developed till 2010. Yet another introductory text book is by professor A.W. Date published by Cambridge University press.

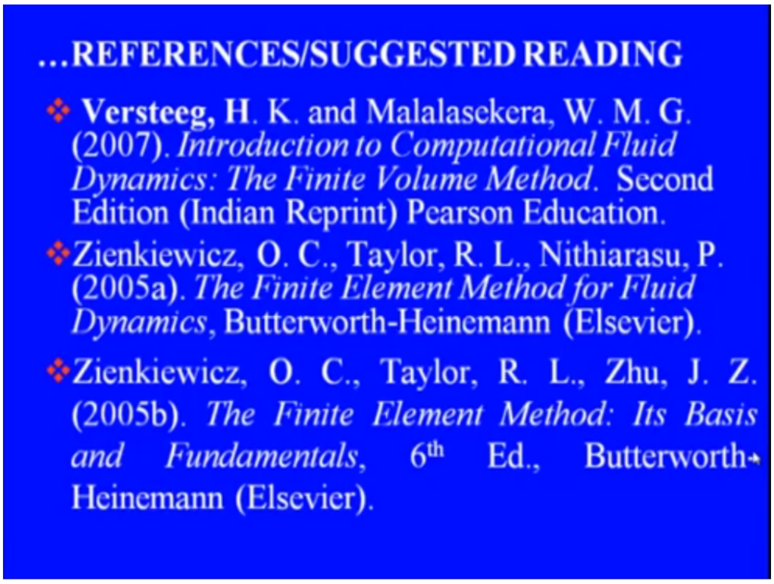
Then similarly computational methods of fluid dynamics by Ferziger and Peric also are very fantastic introductory text. In addition, this book provides you the references of many pertinent differences for complex geometries and turbulence modeling as well.

(Refer Slide Time: 43:57)



If you are interested in finite element method, Reddy's book could be one of them and specifically for fluid applications is the book by Reddy and Gartling is a very good reference. You can also look at the book by professor Sengupta, fundamentals of computational fluid dynamics again the tilt test towards aerospace application in this book.

(Refer Slide Time: 44:18)



Versteeg and Malalasekara's book, this is a very nice introduction for those who are interested in finite volume method. Indent the print was available of this book and similarly for finite element method you can have a look at these 2 classic sets of Zienkiewicz Et al, Zienkiewicz Taylor and Nithiarasu which is finite element method for fluid dynamics and its basics can be covered in yet another volume, which had been referred next.

So these are few popular text books, which I have used and I would recommend you can also have a look at these ones. So this we are going to put a stop to our basic introductions and in the next module we will start up with the first step of the CFD simulation that is mathematical modeling.