**Computational Fluid Dynamics and Heat Transfer**
Prof. Gautam Biswas
Department of Mechanical Engineering
Indian Institute of Technology, Kanpur

Lecture – 01
Historical Perspectives and Introduction to the Course

Good afternoon everyone, it is such a delight to welcome all of you to this course on Computational Fluid Dynamics and Heat Transfer. In the first lecture, we will get introduced to the course and also, we will get introduced to the sources related to learning materials and about the conduct of the course.

(Refer Slide Time: 00:49)
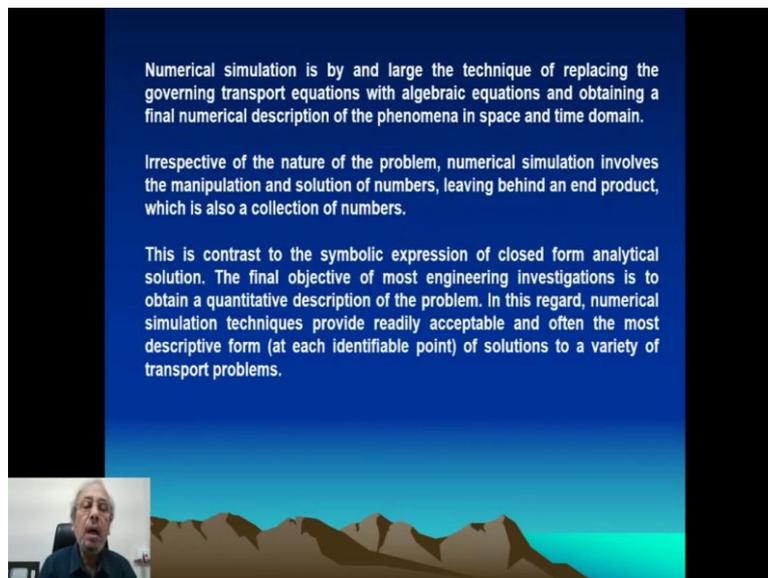
Numerical simulation is by and large the technique of replacing the governing transport equations with algebraic equations and obtaining a final numerical description of the phenomena in space and time domain. Irrespective of the nature of the problem, numerical simulation involves the manipulation and solution of numbers, leaving behind an end product, which is also a collection of numbers.

This is contrast to the symbolic expression of closed form analytical solution. The first objective of most engineering investigation is to obtain a quantitative description of the problem. In this regard, numerical simulation techniques provide readily acceptable and often the most descriptive form of solutions to a variety of transport problems effectively you get to know about the solution at each identifiable point in the domain.

(Refer Slide Time: 02:30)
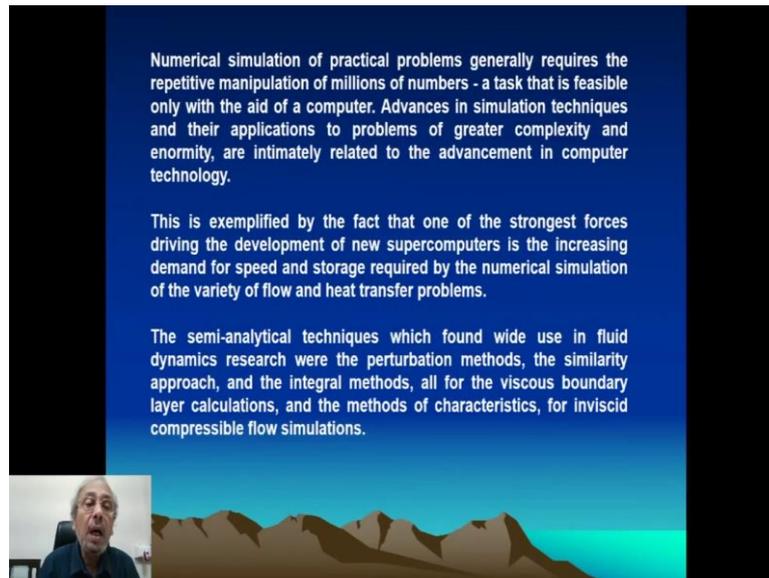
Numerical simulation of practical problems generally requires the repetitive manipulation of millions of numbers – a task that is feasible only with the aid of a computer. Advances in simulation techniques and their applications to problems of greater complexity and enormity, are intimately related to the advancement in computer technology.

This is exemplified by the fact that one of the strongest forces driving the development of the new supercomputers is the increasing demand for speed and storage required by the numerical simulation of variety of flow and heat transfer problems.
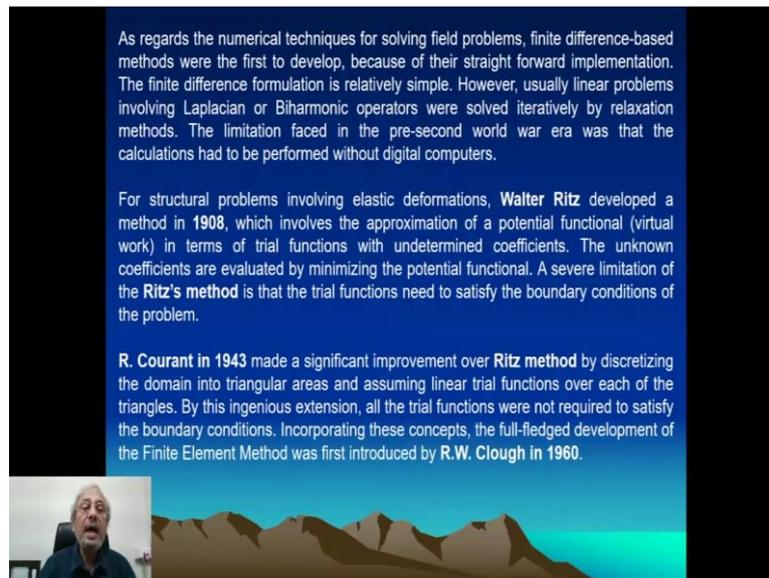
The semi-analytical techniques which found wide use in fluid dynamics research were the perturbation methods, the similarity approach, and the integral methods, all for the viscous boundary layer calculations, and the methods of characteristics, for inviscid compressible flow simulation.

(Refer Slide Time: 04:01)

As regards the numerical techniques for solving field problems, finite difference-based methods were the first to develop, because of their straight forward implementation. The difference formulation is relatively simple. However, using linear problems involving Laplacian or Biharmonic operators were solved iteratively by relaxation methods. The limitation faced in the pre-second World War era was that the calculation had to be performed without digital computers.

The structural problems involving elastic deformation, Walter Ritz developed a method in 1908, which involves the approximation of potential function which is virtual work in terms of trial functions with undetermined coefficients. The unknown coefficients are evaluated by minimizing the potential function. A severe limitation of the Ritz's method is that the trial functions need to satisfy the boundary conditions of the problem.

R. Courant in 1943, made a significant improvement over Ritz method by discretizing the domain into triangular areas and assuming linear trial functions over each of the triangles. By this ingenious extension, all the trial functions were not required to satisfy the boundary conditions. Incorporating these concepts, the full-fledged development of the Finite Element Method was first introduced by R. W. Clough in 1960.

(Refer Slide Time: 06:34)
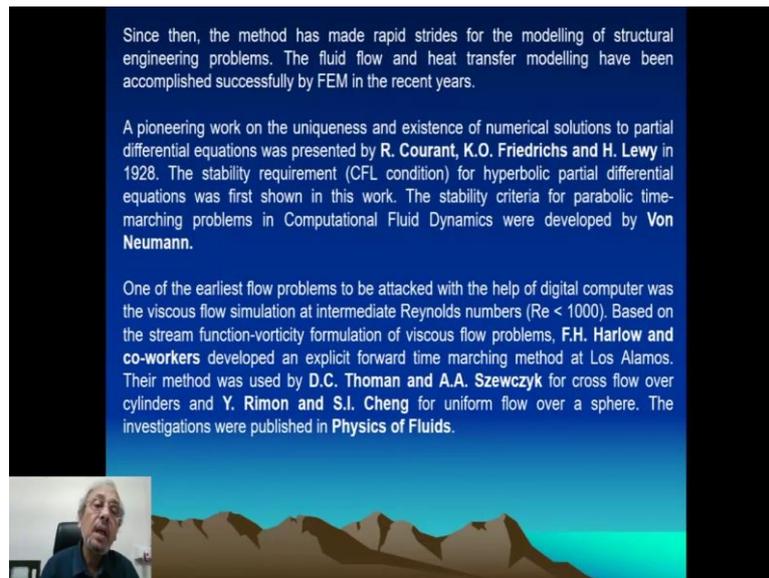
Since then, the method has made rapid strides for the modelling of structural engineering problems. The fluid flow and heat transfer modelling have been accomplished successfully by finite element methods in the recent years.

A pioneering work on the uniqueness and existence of numerical methods to partial differential equations was presented by R. Courant, K. O. Friedrichs and H. Lewy in 1928. The stability requirement which is known as CFL condition for hyperbolic partial differential equations was first shown in this work. The stability criteria for parabolic time-marching problems in Computational Fluid Dynamics were developed by Von Neumann.

One of the earliest flow problems to be attacked with the help of digital computer was the viscous flow simulation at intermediate Reynolds numbers. Based on the stream function-vorticity formulation of viscous flow problems, F. H. Harlow and his co-workers developed an explicit forward time marching method at Los Alamos.

Their method was used by D. C. Thoman and A. A. Szewczyk for cross flow over cylinders and Y. Rimon and S. I. Cheng for uniform flow over a sphere. The investigations were published in a famous journal Physics of Fluids.

(Refer Slide Time: 08:44)
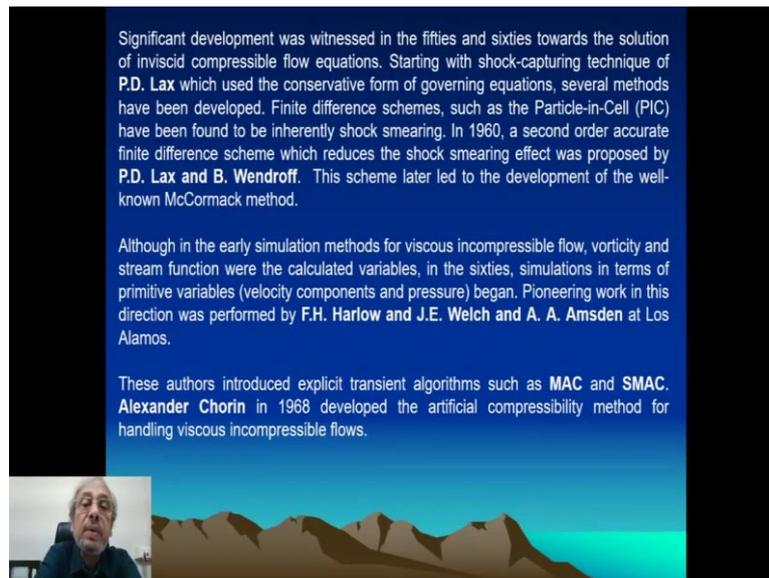
Significant development was witnessed in the fifties and sixties towards the solution of inviscid compressed flow equations. Starting with shock capturing technique of P. D. Lax which used the conservative form of governing equations, several methods have been developed.

Finite difference schemes, such as Particle-in-Cell have been found to be inherently shock smearing. In 1960, a second order accurate finite difference scheme was introduced which reduces shock smearing effect, and this was proposed by again P.D. Lax and Wendroff. The scheme later led to the development of the well-known McCormack method.

Although in the early simulation methods for viscous incompressible flow, vorticity and stream function were the calculated variables, in the sixties, simulations in terms of primitive variables that is velocity components and pressure began. Pioneering work in this direction was performed by again F. H. Harlow, J. E. Welch and A. A. Amsden at Los Alamos.

These authors introduced explicit transient algorithms such as MAC and SMAC, Alexander Chorin in 1968 developed the artificial compressibility method for handling viscous incompressible flows.

(Refer Slide Time: 10:56)
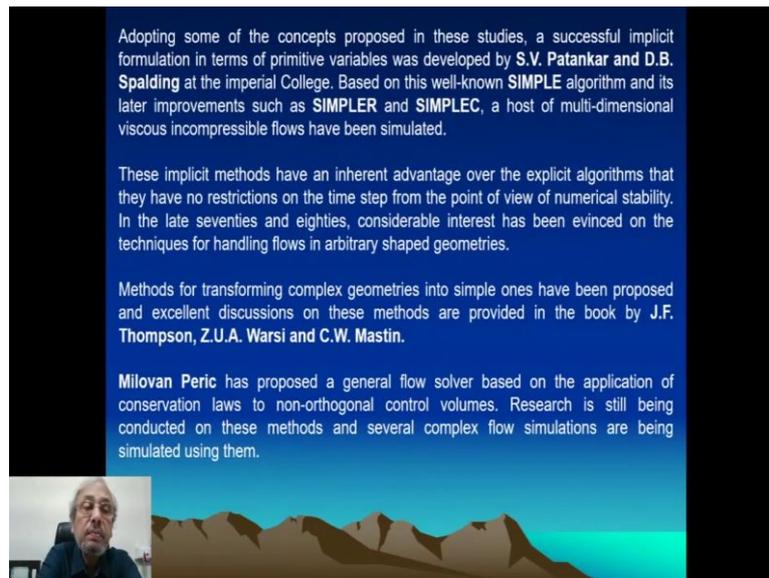
Adopting some of the concept proposed in these studies what I mentioned before, a successful implicit formulation in terms of primitive variables was developed by S. V. Patankar and D. B. Spalding at the Imperial College. Based on this well-known SIMPLE algorithm and its later improvements such as SIMPLER and SIMPLEC, a host of multi-dimensional viscous incompressible flows have been simulated.

These implicit methods have an inherent advantage over the explicit algorithms that they have no restrictions on the time step from the point of view of numerical stability. In the late seventies and eighties, considerable interested has been evinced on the techniques for handling flows in arbitrary shaped geometries.

Methods for transforming complex geometries into simple ones have been proposed and excellent discussions on these methods are provided in the book by J.F. Thompson, Z.U.A. Warsi and C.W. Mastin.

Milovan Peric has proposed a general flow solver based on the application of conservation laws to non-orthogonal control volume. Research is still being conducted on these methods and several complex flow simulations are being simulated using them.

(Refer Slide Time: 13:07)

The role of numerical simulation in engineering is so vital that it has been accepted as an emerging subject itself, which has its own standing based on analytical and experimental knowledge of the cognate engineering discipline and numerical analysis. If we consider advances in fluid mechanical applications, we will observe that major contributions have so far been rendered by a combination of experiments and approximate theoretical analysis.

However, in order to include all the physical details of the problem formulation, total numerical simulation stepped in with its ability to handle the governing equations in their complete form during the end 1960s. Very soon it became a popular and reliable tool in engineering analysis.

Today predictive procedures support experiments, enrich and extend the range of analytical solutions, and finally contribute in product development. Some of these major applications of numerical simulation have been discerned in wind tunnel and combustion studies.

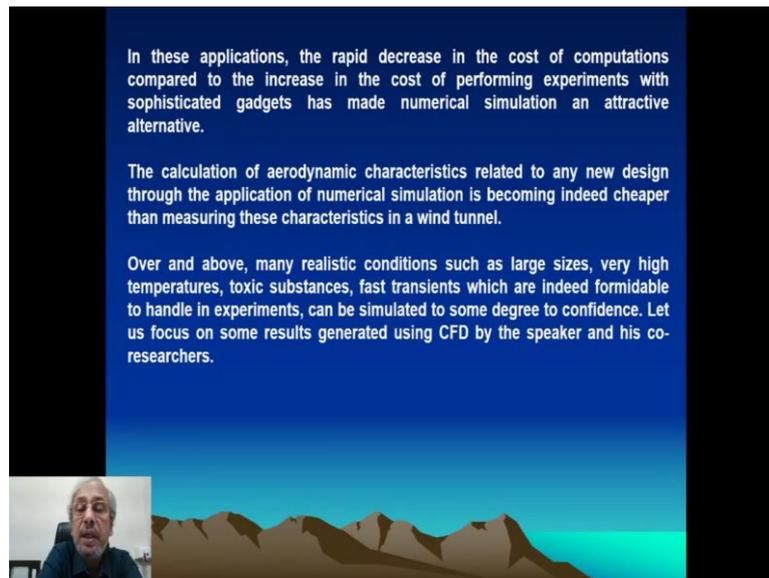(Refer Slide Time: 14:53)

In these applications, the rapid decrease in the cost of computations compared to the increase in cost of performing experiments with sophisticated gadgets has made numerical simulation as attractive alternative.

The calculation of aerodynamic characteristics related to any new design through the application of numerical simulation is been indeed cheaper than measuring these characteristics in a wind tunnel.

Over and above, many realistic conditions such as large sizes, very high temperatures, toxic substances, fast transients which are indeed formidable to handle in experiments, can be simulated to some degree of confidence. Let us focus on some results generated using CFD by the speaker and this co-researcher.

(Refer Slide Time: 16:16)

Ebullition Cycle

Analysis of bubble formation in film boiling. Bubbles are formed periodically (in temporal sense) at the nodes and antinodes on the spatial domain
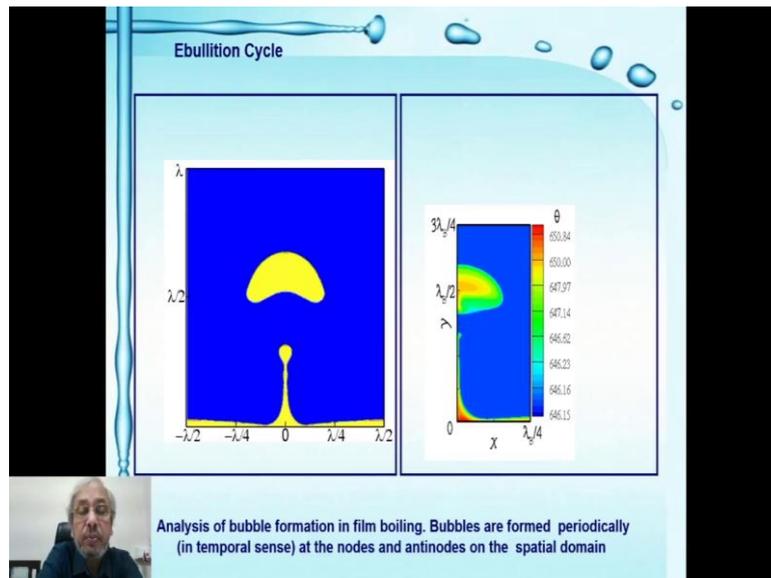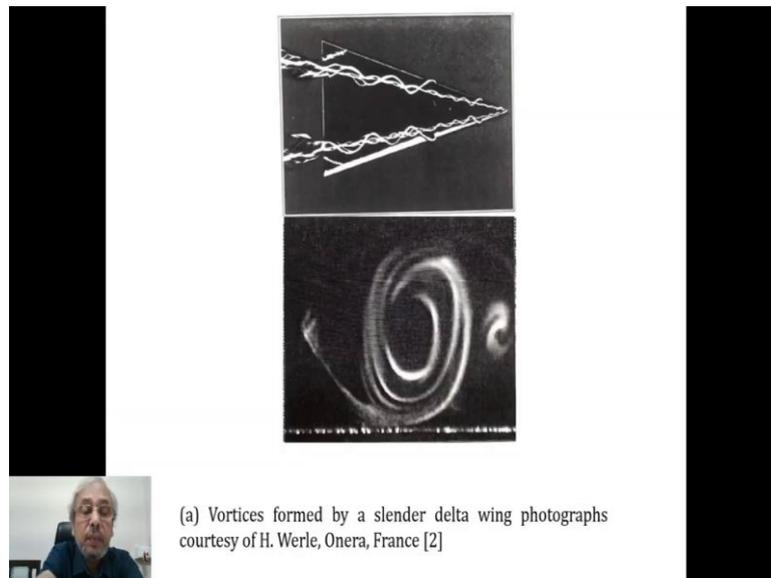
This is basically analysis of bubble formation in film boiling. Bubbles are formed periodically in temporal sense at the nodes and antinodes on the spatial domain.

(Refer Slide Time: 16:36)

(a) Vortices formed by a slender delta wing photographs courtesy of H. Werle, Onera, France [2]

These are the vortices formed by a slender delta wing these photographs were taken in Onera, France. We have done detailed numerical simulations of such vortices.

(Refer Slide Time: 16:56)

(a) Experiment due to Pauley and Eaton (1988 a, b)

Simulation of streamwise longitudinal vortices behand a pair of delta winglets. The counterrotating vortices are seen at different cross planes behind the winglet pair. The flow is turbulent, Reynolds number is 67000.

(b) Present computation

Vector plots of secondary flow: $Re_{H/2} = 67000$
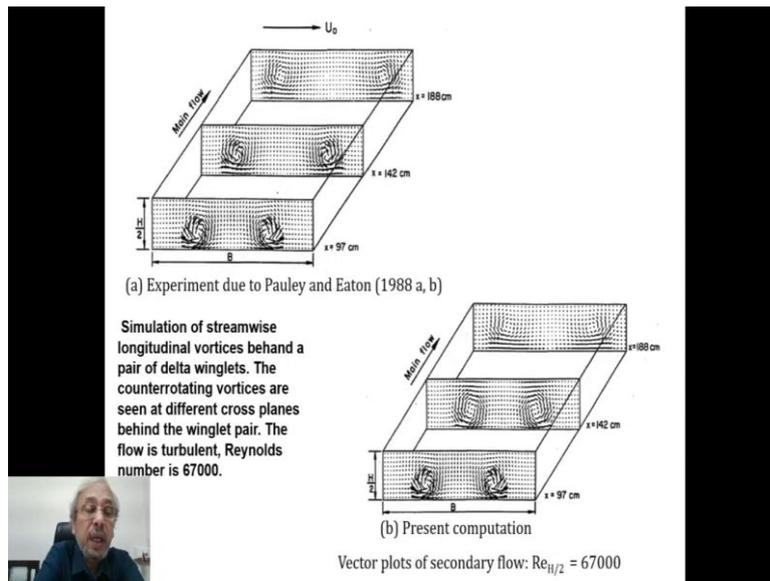
Here you can see one example, simulation of stream wise longitudinal vortices behind a pair of delta winglets. The counterrotating vortices are seen at different cross planes behind the winglet pair.

The flow is turbulent, Reynolds number is 67000 and you can see the experimental result due to Pauley and Eaton experiments were done Stanford University. And you can see numerical simulations which we did at IIT, Kanpur, I mean at the same locations the counterrotating vortices can be designed through these numerical simulations.

(Refer Slide Time: 17:53)

Effect of a falling drop on a Liquid surface. Formation of Secondary and tertiary drops.
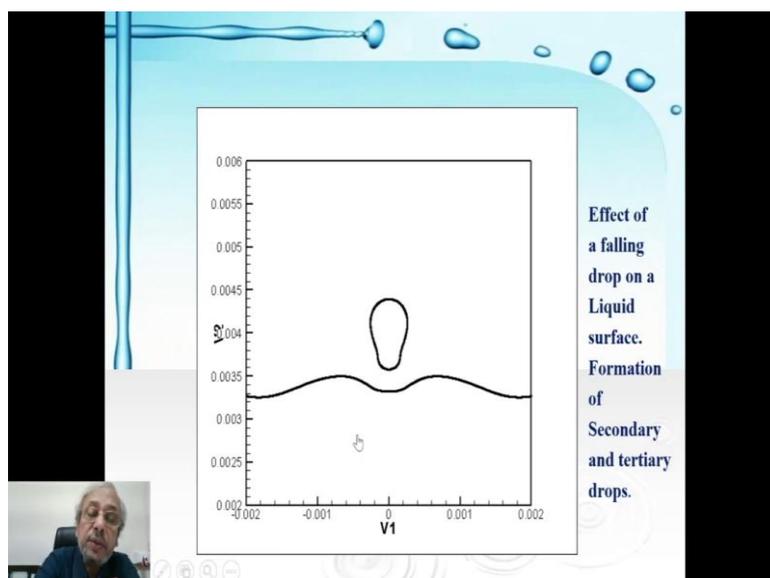
Here we will show effect of a falling drop on a liquid surface, and will be able to appreciate after a falling drop touching a liquid surface how the secondary drop is formed is oscillates forming oblate and prolate shape, again it touches the surface, and tertiary drop is formed. So, these are through direct numerical simulations and we compared our results with very careful done experiments with very high degree of accuracy.

(Refer Slide Time: 18:43)
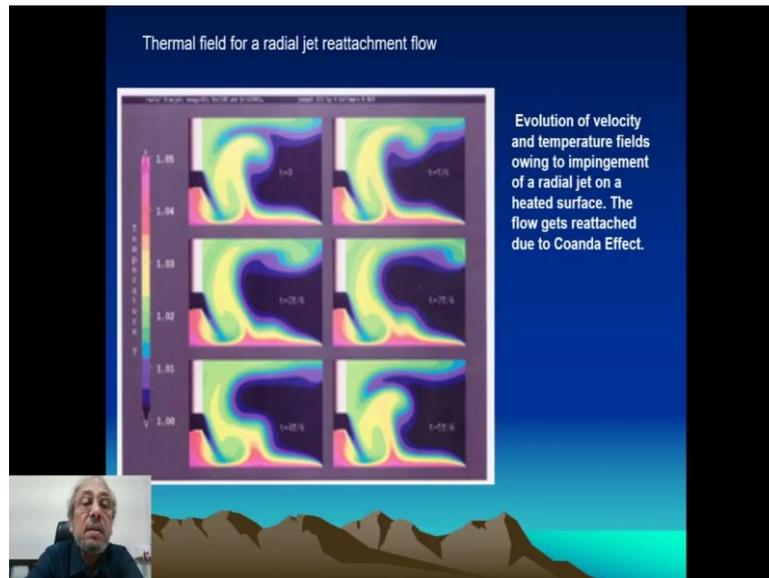
This is another very successful simulation of thermal field for a radial jet reattachment flow. We can see evolution of velocity and temperature fields owing to impingement of a radial jet on the heated surface. The flow gets reattached due to very well-known Coanda effect.

(Refer Slide Time: 19:17)
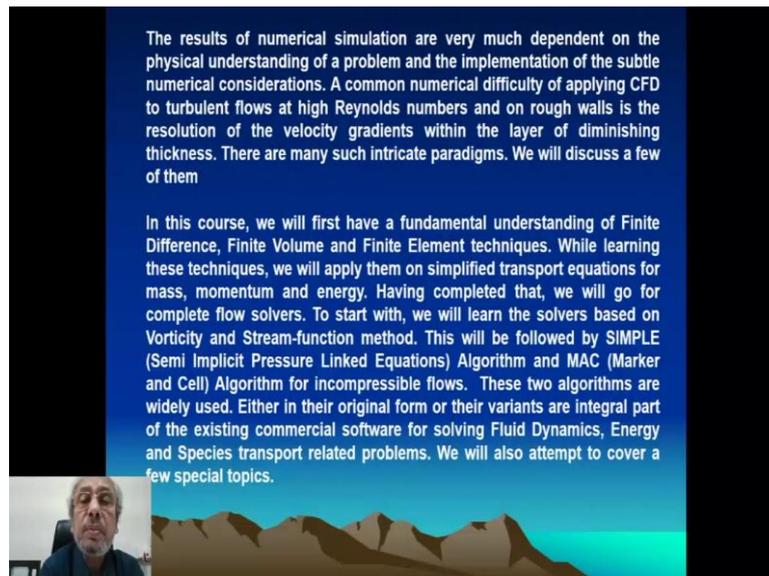
Now, the results of numerical simulations are very much dependent on the physical understanding of a problem, and the implementation of the subtle numerical considerations. I am giving you an example.

A common numerical difficulty for applying CFD to turbulent flows at high Reynolds numbers and on rough walls is the resolution of the velocity gradients within the layer of diminishing thickness. There are many such intricate paradigms, we will discuss a few of them in course of our deliberations.

In this particular course, we will have a fundamental understanding of Finite Difference, Finite Volume and Finite Element techniques. While learning these techniques, we will apply them on simplified transport equations for mass momentum and energy. Having completed that, we will go for complete flow solvers. To start with, we will learn the solvers based on Vorticity and Stream-function method.

This will be followed by SIMPLE, I mentioned about simple algorithm earlier; it is an acronym for Semi Implicit Pressure Linked Equations. So, SIMPLE algorithm and MAC algorithm, MAC is an acronym for Marker and Cell method. We will learn these two incompressible flow solvers; these two algorithms are very widely used. Either in their original form or their variants are integral part of the existing commercial software for solving Fluid Dynamics, Energy and Species transport related problems even combustion related problems. We will also attempt to cover a few special topics.

(Refer Slide Time: 21:44)
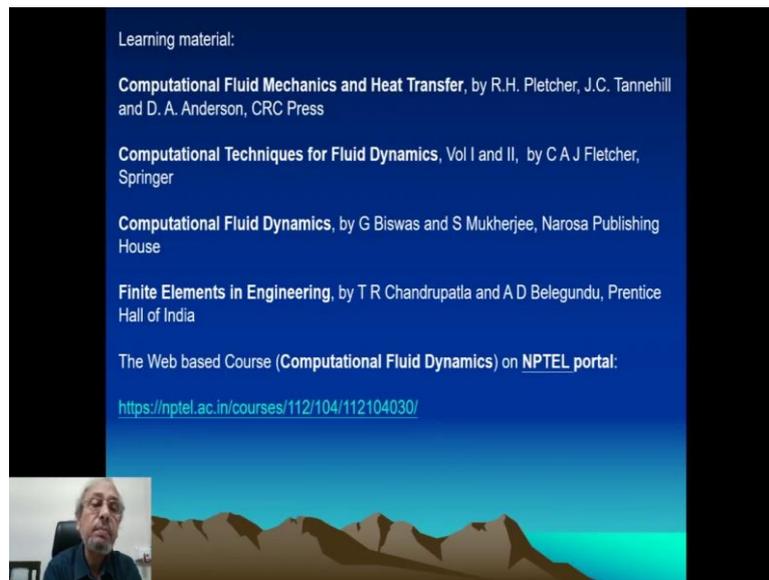
Here are some books. We will follow the lectures, but for the resource materials you know we can follow these books Computational Fluid Mechanics and Heat Transfer by Pletcher, Tannehill and Anderson. Computational Techniques for Fluid Dynamics, Volume I and II, excellent book in two volumes by C A J Fletcher.

Computational Fluid Dynamics by Biswas and Mukherjee; it is an Indian book, Narosa Publishing House publish this, available easily. Finite Elements in Engineering by Chandrupatla and Belegundu; this is very well-known book in finite elements this also available in Indian addition Prentice Hall India publishes it.

In addition, one can always any time refer to web-based course on computational fluid dynamics on NPTEL portal. I have given here the URL (https://nptel.ac.in/courses/112/104/112104302/#). You can copy and paste it on your browser, and can easily go through the learning material.

(Refer Slide Time: 23:10)

Thank you. That is all for today. We will meet again in the next class.