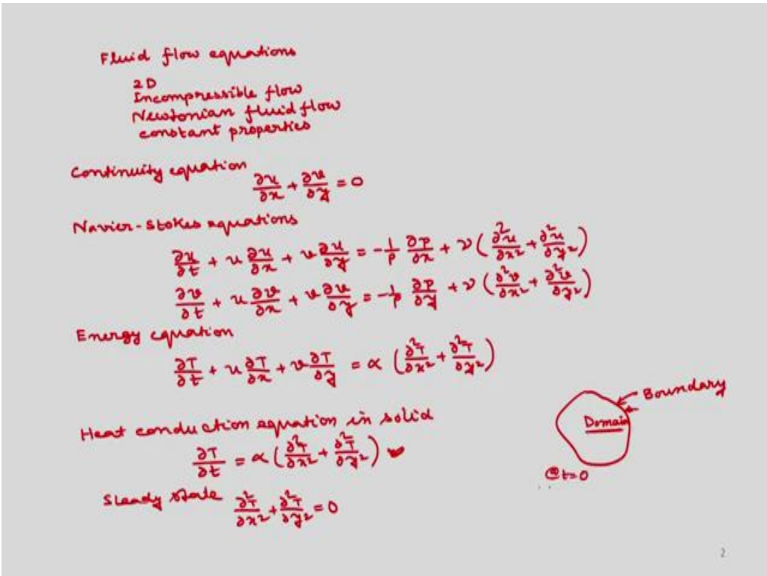
Computational Fluid Dynamic for Incomprehensible Flows
Professor Amaresh Dalai
Department of Mechanical Engineering
Indian Institute of Technology, Guwahati
Lecture 03
Initial and Boundary Conditions

Hello, everyone. So, in last lecture we discussed about basic governing equations of fluid flow, heat transfer, species transport equation and also we discuss about the heat conduction equation in solid. So, these equations we will discretize using different methods in this course, in the domain but at the same time we need to discuss about the initial conditions and boundary conditions.

If the equations are unsteady, so if equations are kind of marching problems, then we need to define the value of any particular variable at the inside the domain at time T is equal to 0, so that is known as initial conditions. And at the same time we need to define the boundary values, so those are known as boundary conditions. So, first let us write the basic governing equations of fluid flow in 2 dimension, incomprehensible and Newtonian fluid flow.

(Refer Slide Time: 01:50)



So, if you write the fluid flow equation, which we discuss in last class, so for 2D, incomprehensible flow Newtonian fluid flow and constant properties. With negligible body force and heat generation. So, with that if you write then you will get del u by del t let us write first continuity equation, so first let us write the continuity equation.

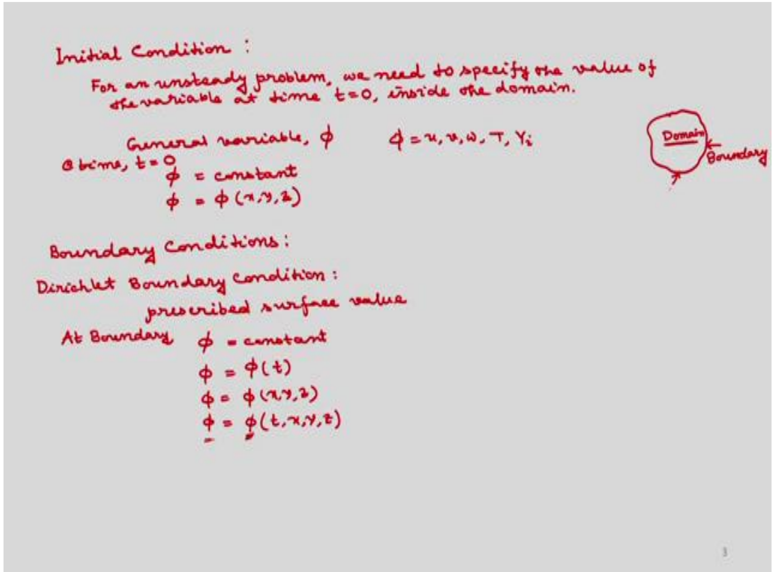
So, this is your del u by del x plus del u by del y is equal to 0 for incomprehensible fluid flow and the Navier Stokes equation, which is momentum equations, so this is your x momentum equation for constant properties, so there is a pressure variant del p by del x plus kinematic viscosity Nu del 2u by del x square plus del 2u by del y square, so this is in 2 dimensions. And y momentum equations similarly you can write as del v by del t plus u del v by del x plus v del v by del y is equal to minus 1 by rho del p by del y plus Nu del 2u by del x square (plus) sorry del 2v, del 2v where square del 2v by del y square.

And energy equation if you write then it will be del t by del t plus u del t by del x plus v del t by del y is equal to alpha which is your thermal diffusivity k by rho cp del 2t by del x square plus del 2t by del y square with negligible heat generation. So, these are the equations we will use in this course as well as for the solid if you write the heat conduction equation, then if you put the velocity is 0 then you can write the heat conduction equation.

Heat conduction equation in solid, so you can write del T by del t is equal to alpha del 2T by del x square plus del 2T by del y square. And if you have the steady state assumptions ,then the temporary term will go to 0, so we can write, for steady state you can write del 2T by del x square plus del 2T by del y square is equal to 0. So, this governing equations will discretize and using different method and we will solve in the interior domain.

So, if you have this is the domain and this is the boundary, so discretize equation you will solve inside the domain but you need to specify the value of that particular variable at the boundary, so that is known as boundary conditions. And if it is unsteady problem, so it is marching in time, so for that you need to specify the variable at p is equal to 0, so that is known as initial conditions. So, let us discuss about initial condition and boundary conditions.

(Refer Slide Time: 06:15)



So, initial condition. So, as we discussed for an unsteady problem, we need to specify the value of the variable at time t is equal to 0 inside the domain. So, if this is your domain, so this is your interior points, interior points and this is your boundary. So, you have to specify the value inside the domain at time is t is equal to 0 that is known as initial condition. So, there are different equations we have already discussed, so the particular variable you need to specify at t is equal to 0 at inside domain.

So, we will write these initial condition for any general variable phi, so general variable phi if you write, so phi may be your u, v, w or any other temperature or any species, so for any variable, general variable phi, so phi may be at the time t is equal to 0 at time t is equal to 0, phi may be constant, you can specify at constant value at the domain or phi may be function of x, y, z means especially varying.

So, phi may vary especially or it may have some constant value and that you need to specify at time t is equal 0 to inside the domain. So, that is known as initial condition. Now, as I told initial condition you need to specify only for unsteady problem later we will see that these are marching problems, so for marching problems you need to define the initial condition or prescribe the initial condition at the interior domain.

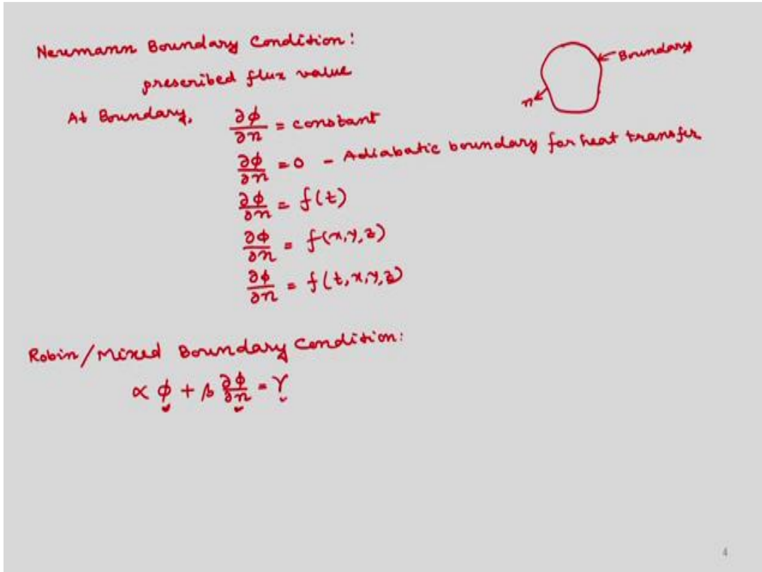
At the same time you need to specify the value of the variable at the boundary, because you will solve the discretize algebraic equations inside the domain, so you need to specify the value of any variable at the boundary. So, those that is known as boundary conditions. So, now we will discuss about the boundary conditions. So, we will first categorize in 3 different types of boundary conditions, one is Dirichlet condition, then Neumann condition, then Mixed or Robin condition.

So, Dirichlet condition first let us discuss, Dirichlet boundary condition. So, Dirichlet boundary conditions, in Dirichlet boundary condition you need to specify the value of the variable at the boundary, that value may be constant, it may be temporarily varying or it may be especially varying. So, that means you have prescribed surface value, so the value is specified at the boundary, value is specified at the boundary, so that is known as Dirichlet boundary condition.

And for any general variable phi, so it may be constant, at the boundary, then it may be temporarily varying, it may be function of times, so it may be temporarily varying, it may especially varying, so phi is function of x, y, z or it may vary both in time and space. So, it may vary with specially and temporarily, So, what it is, so you are specifying the value at the boundary, so that is known as Dirichlet type boundary conditions.

Next we will discuss the Neumann type boundary conditions, in Neumann type of boundary conditions you need to specify the gradient, so the gradient is prescribed at the surface or boundary.

(Refer Slide Time: 11:34)



So, now we will discuss about Neumann boundary condition, so you have prescribed flux value, prescribed flux value, that means gradient is specified at the boundary. So, this is your boundary, so gradient is specified at the boundary. So, at boundary in Neumann type boundary conditions, it may be that the gradient whatever you are specifying, so that will be always flux will be the normal to the surface.

So if it is normal to the surface then del phi by del n will be constant, so you can specify a constant value of that flux that means the gradient value is constant or it may be 0, so for a particular condition like adiabatic condition in heat transfer, where there will be no heat flux from the boundaries, so that means your del t by del n is equal to 0. So, that is Neumann type boundary conditions, adiabatic type boundary conditions.

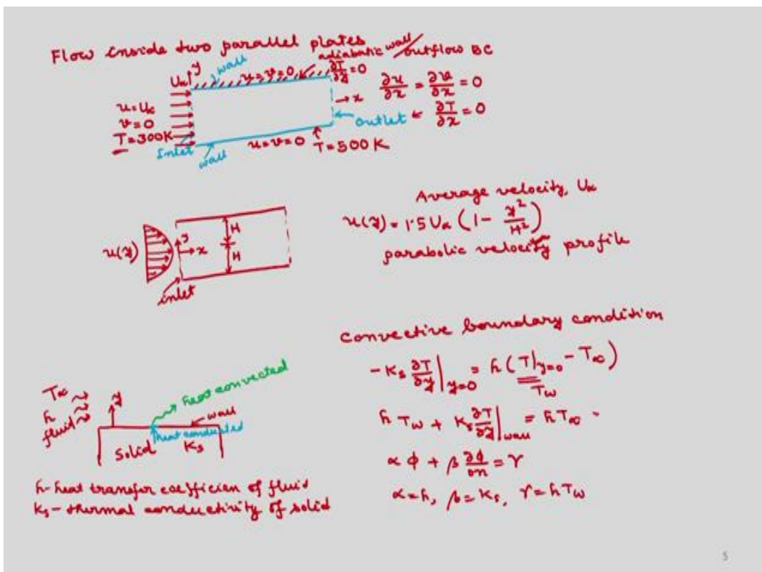
So, that means del phi by del n may be 0, constant value may be 0 for adiabatic boundary, for heat transfer, you will get also I will show for a fluid flow problem also the normal gradient will be 0, or del phi by del n may be function of time, it may be function of space as we discussed and it may be function of both space and time.

So, the boundary conditions you are specifying in terms of flux, so flux may be constant or that flux may be vary with time or flux may vary with space or flux may vary with both time and space. So, that is known as Neumann type boundary conditions. Now, we will discuss about

Mixed or Robin boundary condition. So, in Robin or mixed type boundary conditions it is combination of these two, Dirichlet and Neumann type boundary conditions.

So, it will value plus the gradient will be specified such a way that it will have both type of boundary conditions. So, in general we will write this as alpha, alpha is phi plus beta del phi by del n, it is normal flux is equal to gamma. So, you can see that phi is a constant value, so it is Dirichlet type, del phi by del n is the flux or gradients, so that is specified equal to some value gamma. So, we will take some examples and will show these different types of boundary conditions.

(Refer Slide Time: 15:17)



First, consider flow inside 2 parallel plates, infinite parallel plates, so flow inside or flow through or flow inside 2 parallel plates, flow inside 2 parallel plates. So, first you need to find what is your computational domain. So, you have 2 parallel plates, so now this is your boundary then, how many boundaries are there in 2 dimensional situations? You have 4 boundaries. Now, as you are solving a fluid flow and heat transfer problem then you need to solve the governing equations, nernst of equation and the energy equation in 2 dimension what we wrote in todays class in the beginning.

Now, you need to specify the boundary conditions. So, this is your flow inlet, flow is coming and entering here, flow is going out through this boundary, so it is outlet and these are wall. So, now the walls boundary conditions let us discuss. So, in the wall we know that we will have no slip boundary conditions that means the fluid particles sitting on the solid wall, so it will have the same velocity at the wall.

So, if the continuum hypothesis is valid, then obviously here you will have the no slip boundary conditions and in that case the velocity is at the boundary you need to specify a 0. So, velocity is if in 2 dimensions u and v, so u v will be 0 at the solid wall and these are Dirichlet type boundary conditions, because you are specifying the value of the velocity at the wall.

So, that means your u is equal to v is equal to 0 and u is equal to v is equal to 0. At the inlet if you specify the velocity, then let us say that you are specifying a velocity uniform velocity at the inlet, so it is entering let us say it is u infinity, so that means u is equal to u infinity you are specifying and b is equal to 0. So, that means you are specifying the value of the velocities, so this are Dirichlet type boundary conditions.

And at the outlet generally we assume that it is a fully developed conditions that means there will be no axial variation of this variables, that means, if it is x direction, then you will have del u by del x is equal to del b by del x is equal to 0, so that means axial gradient of this velocities are 0. So, this is outflow boundary conditions, outflow boundary conditions. So, obviously you can see that we are specifying the flux at the outlet that means you are actually specifying the normal gradient. So, obviously these are Neumann type boundary conditions.

Now, if you are solving with heat transfer, then you need to specify the temperature at the inlet as well at the valve. Let, us say that valve temperature is higher than the inlet temperature, so if it is if you are specifying T let us say it is maybe 500 degree centigrade, this is also, let us say adiabatic wall, there is no heat loss from this boundary, no heat loss from the boundary, so this is adiabatic wall, so no, so that means no heat transfer across this wall, so obviously it will be del T by if it is normal direction is y then del t del y is equal to 0.

So, you can see that it is Neumann type boundary condition and on the bottom of wall we are specified the temperature it is Dirichlet type boundary condition and at the inlet we will specify temperature let us say T is equal to, let us say some 300 not degree centigrade let us say it is kelvin, 500 kelvin and 300 kelvin, 500 kelvin, 300 kelvin. So, this is your specifying the value of the temperature at the inlet, so obviously it is Dirichlet type boundary condition.

And at the outlet similar way, you can define del T by del x equal to 0, this is the boundary conditions, for this flow inside to parallel plates. Again you may need to specify the inlet velocity as parabolic, where especially it will vary. So, if want to give the fully developed condition at the wall, then you need to specify parabolic boundary condition. So, this you can see that it will especially vary.

So, for a particular problem let us say flow inside parallel plates, so this is the inlet, this is the inlet and this is your x from the center and this is your y and let us say the height is H and this is also H. So, the distance between two parallel plates is 2H, so this is x y, so you can specify the parabolic profile like this, so at the center you will have maximum, so you can see here the velocity is varying especially.

So, u is function of y .And this u function of wall you can specify as let us say if average velocity is u infinity, average velocity is u infinity then 1.5 u infinity into 1 minus y square by H square, so this you have derived analytically for a fully developed flow problem that is known as (())(22:32) flow. So, that we are gaining as inlet at the you are giving this as inlet condition. So, we can see y is equal to 0, 1.5 to infinity so maximum velocity is your 1.5 times the average velocity.

And at y is equal to H, y is equal to H, u is equal to 0, y is equal to minus H, u is equal to 0 so this is you parabolic profile, parabolic velocity profile. And this is Dirichlet type boundary conditions, this is Dirichlet type boundary conditions, but it is especially varying, u is function of y. Now, let us have one example of Robin boundary condition. So, let us consider a solid, here, this solid, over it there is some fluid flow, so you have temperature T infinity and H is the heat transfer coefficient, H is the heat transfer coefficient of the fluid, heat transfer coefficient of fluid.

And for solid k is the thermal conductivity. Let us say k s, so k is the thermal conductivity of solid. So, what is happening? So, we have the solid, where the surface you have a fluid flow that fluid temperature is t infinity and the heat transfer coefficient is H. So, we can see that this fluid will take away the heat from the solid. So, at the solid wall you can have the energy balance and that energy balance if you do, so at the solid wall you can have the energy balance.

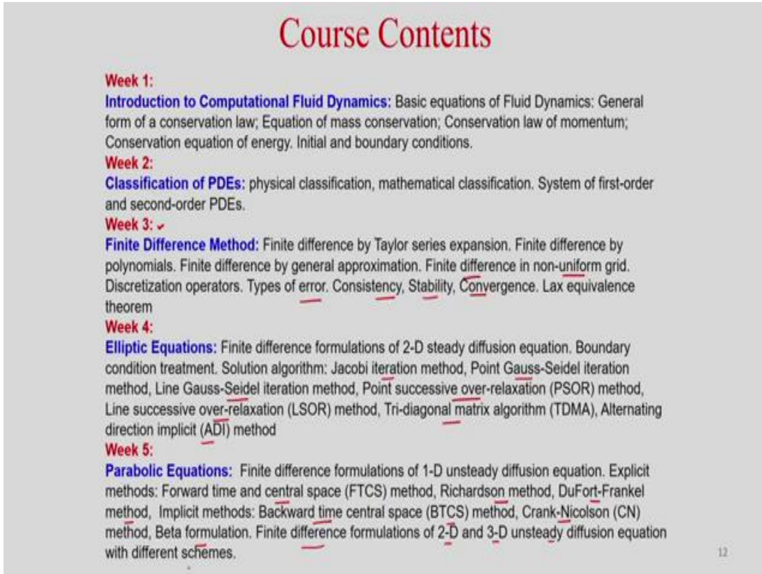
So whatever heat is conducted, heat is conducted, so that is actually convicted, heat convicted because you have a fluid flow, so heat is convicted, so this is the energy balance if you do that so we can see, so this boundary condition is known as convective boundary condition, convective boundary condition. So, now we are doing the energy balance heat conducted, what is heat conducted? So, this is your y then you can write minus k s del T by del n or y you can write it is normal, y is the normal to the surface del T by del y at y is equal to 0.

So, that is the heat conduction taking place and at y is equal to 0 we are finding what is the heat flux and equal to now whatever heat is conducted, so if you have a heat transfer coefficient H and the boundary temperature t at y is equal to 0 minus t infinity, so that is Newton law of cooling, right so it is Fourier law and this is Newton law of cooling. So, h is so T at y is equal to 0 minus T infinity, right.

So, you can see now if you see so you can rearrange it and you can see that it will be h if y is equal to 0 it is let us say wall, so it is Tw if you write, so then it will be h Tw plus or ks del T by del y at wall is equal to, so this will go this side so it will be h into t infinity. So, you can see that this is if you recall we have written alpha phi plus beta del phi by del n is equal to gamma, so this is the Robin boundary condition, combination of Dirichlet and Neumann type boundary condition.

So, here you can see the alpha is equal to h, beta is equal to ks and gamma is equal to hTw. So, these boundary condition you can see one of the example, examples of Robin type of boundary conditions and in heat transfer convective boundary condition is Robin type boundary condition. So, that we have shown here.

(Refer Slide Time: 27:25)



So, now let us discuss about the course contents. So, already we have discussed this week 1 lectures, so we have already in the first lecture we have shown the application of CFD and why we need to study the CFD that we have discussed.

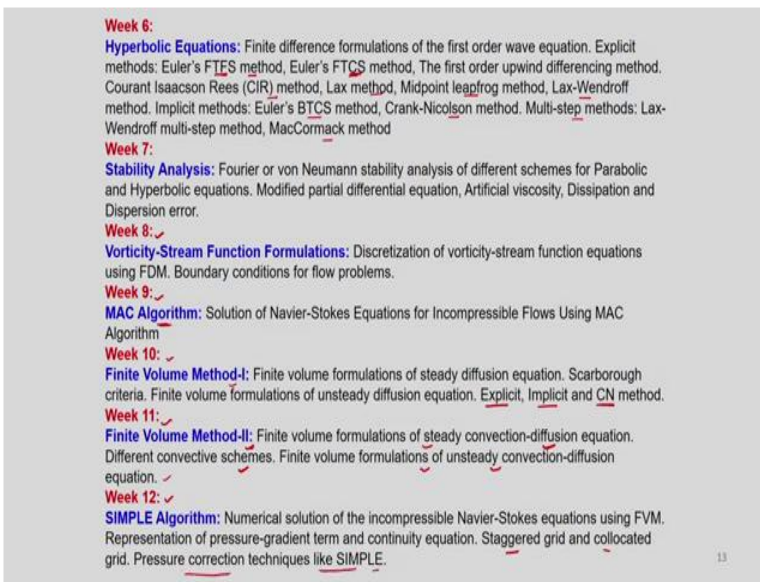
In second lecture we have discussed about the governing equations, fluid flow as well as heat transfer and next in todays lecture we discuss about the initial condition and boundary condition. In week 2, we will classify the PDEs, so we can classify the PDEs in two ways, mathematically and physically that we will discuss and will take some examples and will show that which type of PDEs these are.

Next week then week 3, in week 3 we will discuss about the finite difference method. So, will introduce finite difference method in week 3 and first will use Taylor series expansion to find the finite difference of any gradient and will also use different techniques in finite difference method one is Taylor series then another is polynomial and another is general approximation. And also will do finite difference in no-uniform grid and will discuss type of errors consistency, stability and convergence.

Next in week, 4 will consider only elliptic equation. So, steady heat diffusion equation is example of elliptic equation that we will consider and will discretize using finite difference method and will learn different discretization schemes. So, then will solve this discretize equation using Jacobi iteration method, Point Gauss-Seidel method, Line Gauss-Seidel iteration method, point successive over-relaxation method, Line successive over-relaxation method and Tri-diagonal matrix, Algorithm TDMA and alternating direction implicit method.

In week 5 we will consider parabolic equation, we will take one model equation, unsteady 1 dimensional heat diffusion equation and will discretize this equation using difference scheme both explicit and implicit. So, we can see forward time and central space, Richardson method, DuFort-Frankel method, in implicit methods we will see backward time, central space BTCS, Crank-Nicolson, Beta formulation and also will see in finite difference formulation about 2D and 3D unsteady diffusion equation.

(Refer Slide Time: 30:23)



In week 6 we will consider hyperbolic equation, so we will consider 1 dimensional wave equation and will learn different discretization scheme, you can see here Euler FTFS method forward time and forward space, Euler FTCS forward time central space, then CIR method, Lax method, Midpoint leapfrog method, Lax-Wendroff method and in implicit method will learn Euler BTCS method, Crank-Nicolson and Multi-step method like Lax-Wendroff and MacCormack method.

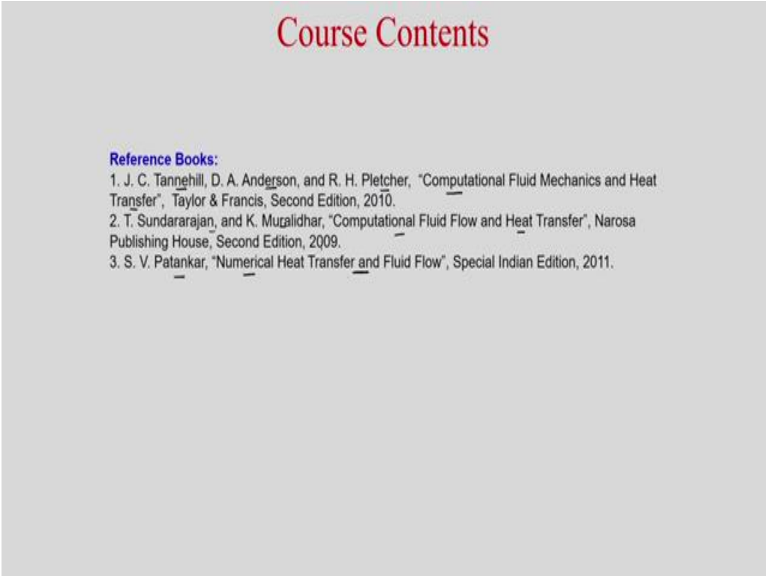
Then week 7, will learn the stability analysis, will study here only Von Neumann stability analysis and will consider this parabolic and hyperbolic equation and will find what is the stability criteria using Von Neumann stability analysis. In week 8, will consider Vorticity steam function equation and that will solve using finite difference method and will consider two problems, leap driven cavity problem and flow inside two parallel plates and will discuss about the boundary conditions.

Week 9, we will solve the full Navier Stokes equation using finite difference method algorithm MAC will use to discretize this Navier Stokes equation, MAC stands for Marker and Cell method. In week 10, will start finite volume method, so first will study the basics of finite volume method, then will consider steady heat diffusion equation or steady diffusion equation and then unsteady diffusion equation and will discretize these equations using finite volume method.

And will discretize using both explicit, implicit method and also will learn Crank-Nicolson method. In week, 11 will continue with the finite volume method and first we will discretize steady, convective, diffusive equation using finite volume method and we will discuss about different convective schemes and after that will solve this finite volume then will solve the unsteady convective diffusive equation using finite volume method.

In last week, week 12 we will solve the full Navier Stokes equation using finite volume method. So, we will considered steady state 2 dimensional Navier Stokes equation and we will solve using simple algorithm. And also will discuss about the Staggered grid and collocated grid in this lecture and also will derive the pressure correction equation and will discuss about the pressure correction techniques like SIMPLE.

(Refer Slide Time: 33:23)



In this course mostly we will follow these 3 books, one is Computational Fluid Mechanics and Heat Transfer by Tannehill Anderson and Plectcher, then Computational Fluid Flow and Heat Transfer by Sundararajan and Muralidhar and these 2 books are mainly for finite difference method and for finite volume method you may refer these books Numerical Heat Transfer and Fluid Flow by S.V Patankar, it is very popular book for finite volume method and Indian version is available you can purchase these books for your reference.

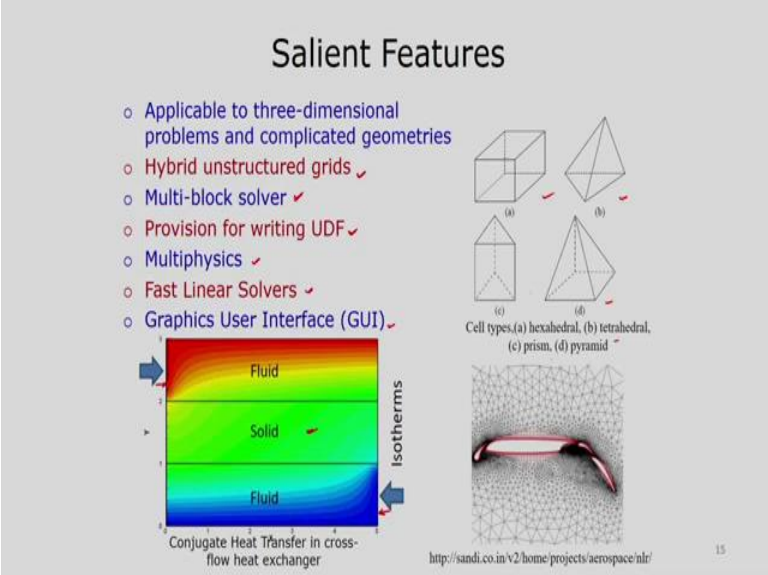
But you need to follow my class note, because few derivations will derive from other differences books, which I have not noted here. And now we will show some CFD results from the in house code developed at ITG by the PHD students and M.Tech students. So, this is some motivation to you that students you can write some programs or using CFD for fluid flow and heat transfer problems. CFD if you want to learn then only theory is not sufficient, you need to write the computer code then only you will learn CFD. So, today now I will show some results from the ANU solver.

(Refer Slide Time: 35:08)

So, this ANU solver is known as AnuPravaha this project is funded by DAE Department of Atomic Energy, BRNS Government of India, we have developed a general purpose CDF solver for fluid flow problems over a hybrid unstructured grid, this is a multi-physics problem so here you can solve different types of problem. And these solver is developed particularly by 4 PHD students and more than 35 M.tech students and several project staffs.

So, whatever results I am showing here, so this are actually solved using this ANU solver AnuPravaha. So AnuPravaha means ANU is atom and Pravaha means flow, so these name was given from BRNS because this is department of atomic energy, so with that the name is given as AnuPravaha.

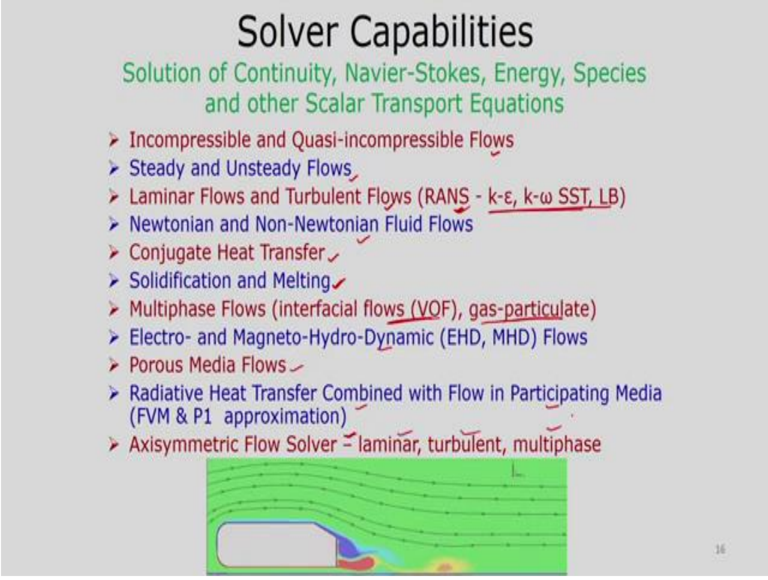
(Refer Slide Time: 36:12)



The salient features are that it is inherently 3 dimensional code, you can generate the unstructured grid in a complicated domain. And we have hybrid unstructured grid, so 4 types of grid we have like hexahedral, tetrahedral, prism and pyramid in our solver, it is multi-block solver, so you can solve for more than 1 free zone and more than 1 solid zone you can see that it is a heat exchanger problem, so fluid is coming in here and another fluid is coming in here, so it is a hot fluid, is a cold fluid and there is a solid.

So, you have 2 fluid zone, 1 solid zone, so this type of problem you can solve using multi-block solver. And we have also prohibition for writing user defined function, we have multi-physics problem and we have used first linear solvers and we have also graphical user interface.

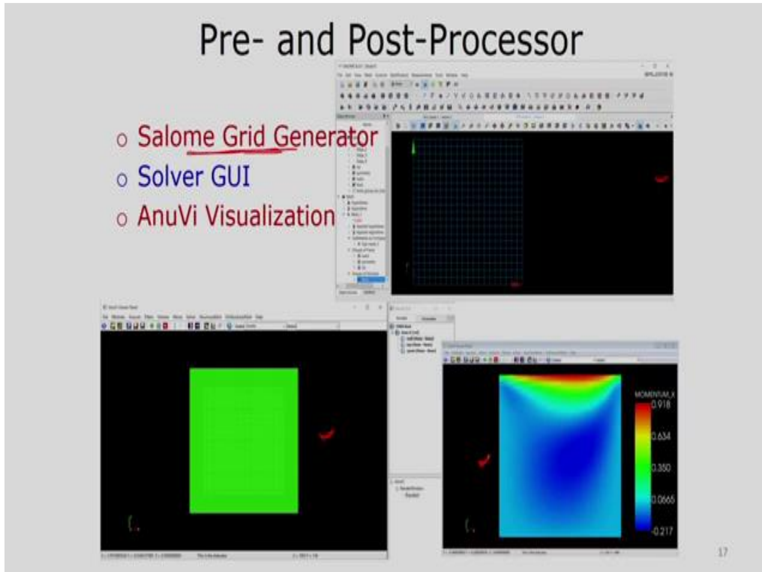
(Refer Slide Time: 36:58)



So, now we will discuss about what are the capabilities of this solver, so we have incompressible and quasi-incompressible flow, steady and unsteady flows, Laminar flows and Turbulent flows. In turbulent flows we solve Reynolds-averaged Navier Stokes equation using different 2 equation models, we also can solve Newtonian and Non-Newtonian fluid flows, conjugate heat transfer, phase change like solidification and melting.

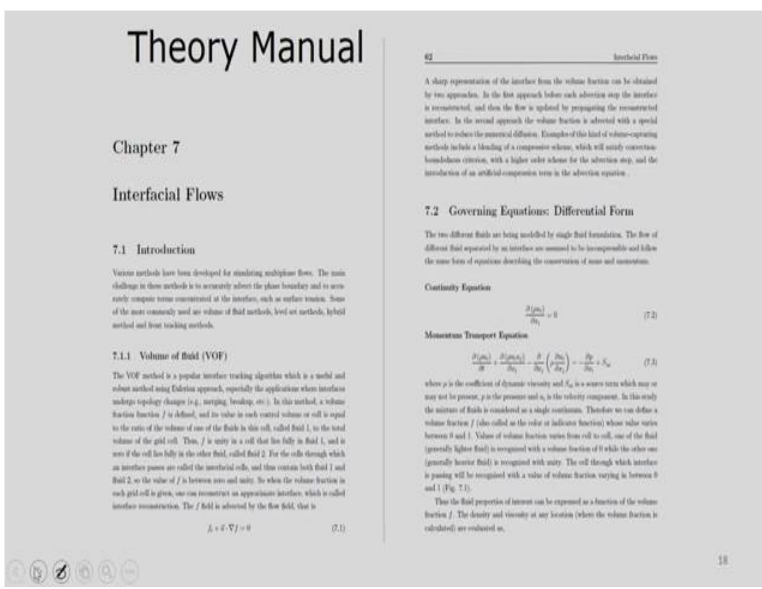
Multi-phase flows for liquid and airflows way of inter-facial flows so for inter-facial flows, we use volume of fluid method and for gas and particle flows, we have gas particulate flow solver. We have also electro and magneto-hydro dynamic flow solver, Porous media solver, Radiative heat transfer combined with flow in participating media and also we have developed some axisymmetric flows solver for laminar, turbulent and multi-phase. So, you can see that it is a multi-physics solver where you can solve different types of problem.

(Refer Slide Time: 38:03)



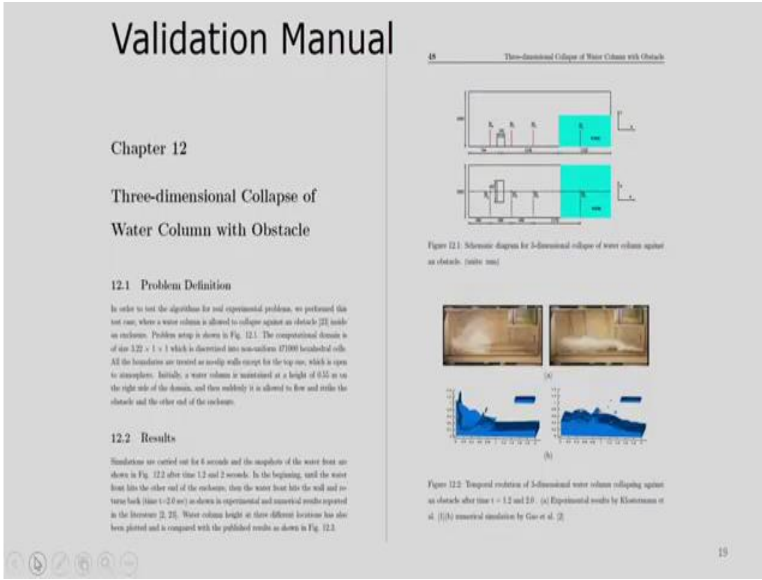
So, these are some snapshots of pre-processors Salome is a grid generator, this is actually open source grid generator and this is the GUI of that, this is the solver GUI and this is the AnuVI GUI, so this is actually post processing solver developed at BARC Bhabha Atomic Research Center at DAE institute.

(Refer Slide Time: 38:23)



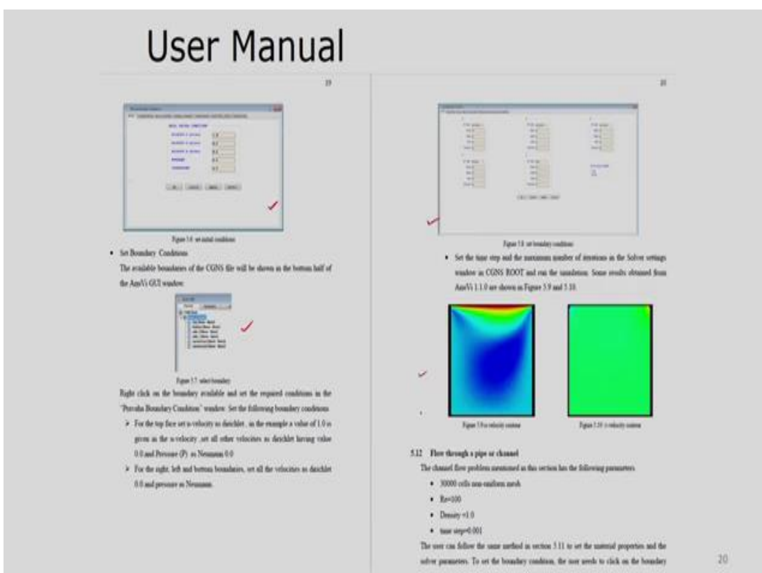
We have also developed over the period theory manual, so that one can learn the theory of each module.

(Refer Slide Time: 38:33)



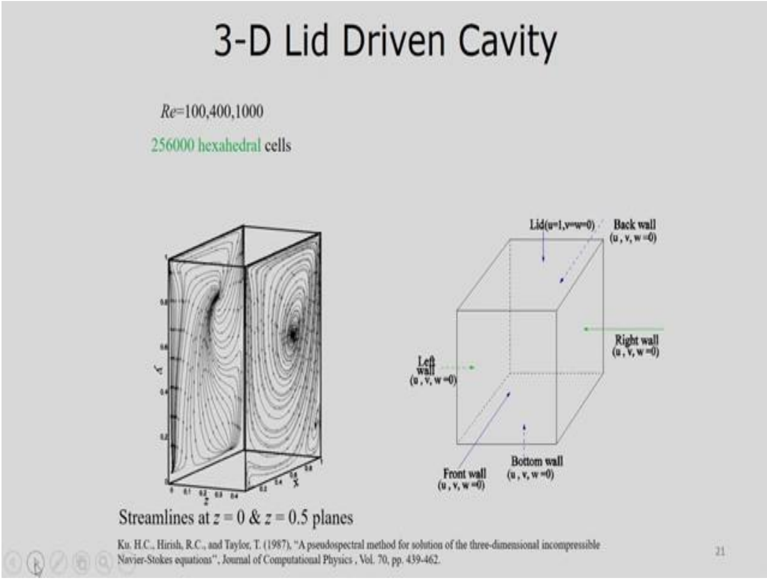
We have validation manual so whatever problem we have solved and validated with the results available in the literature either experiment or numerical results, so that we have put in the validation manual.

(Refer Slide Time: 38:47)



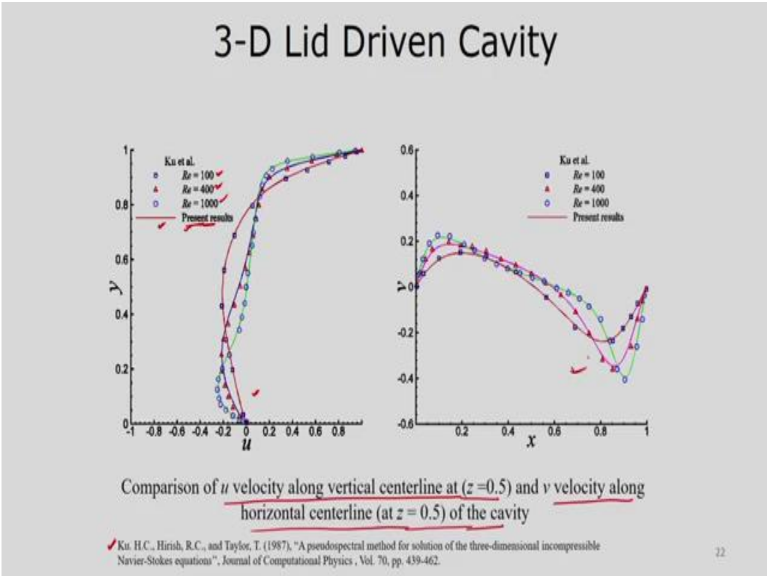
And we have user manual, where one can use it through the GUI. So, you can see how to give the inputs in GUI, so that we have described and how to post-process it.

(Refer Slide Time: 38:59)



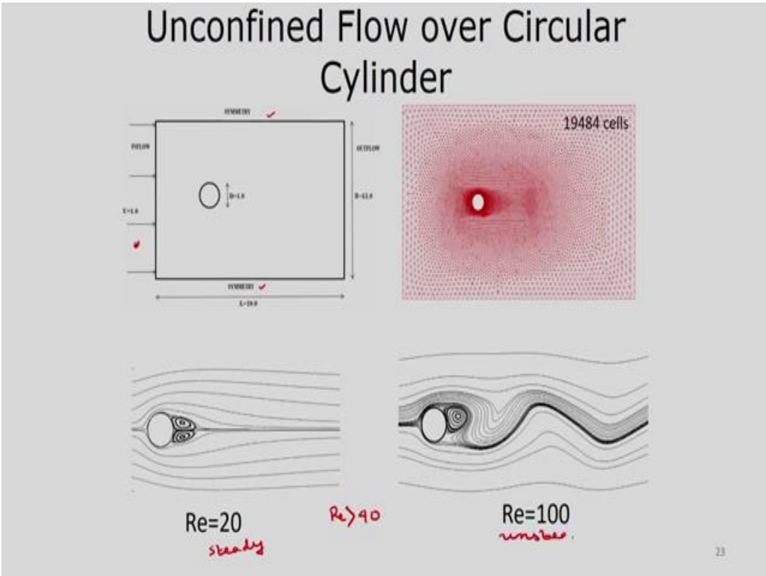
Now, let us see some validation results. So, first we will discuss 3 dimensional Lid driven cavity problems, so it is a famous problem, so if someone is writing his own solver then this is the easiest problem to solve that whether your code is cut or not. So, in 2 dimension you have 2 dimensional Lid driven cavity problem, here it is 3 dimensional lid driven cavity problem, you can see that upper lid is moving with velocity 1 and other all the walls velocities are 0. So, you will get these are the steam line profiles at different z locations, z is equal to 0 and z is equal 0 point 5, 0 point5 is the mid-plane.

(Refer Slide Time: 39:48)



And these results if you can see that u velocity along vertical central line at z is equal to 0 point 5 we have compared here, so you can see our result is with solid line and from the literature Ku-et-al the reference is given here. So, for different Reynolds number 100, 400, 1000 our present results is matching avail with the Ku-et-al results. This is the plot of V velocity along horizontal center line at z is equal to 0 point 5 of the cavity and you can see the velocity profile is matching avail with the literature.

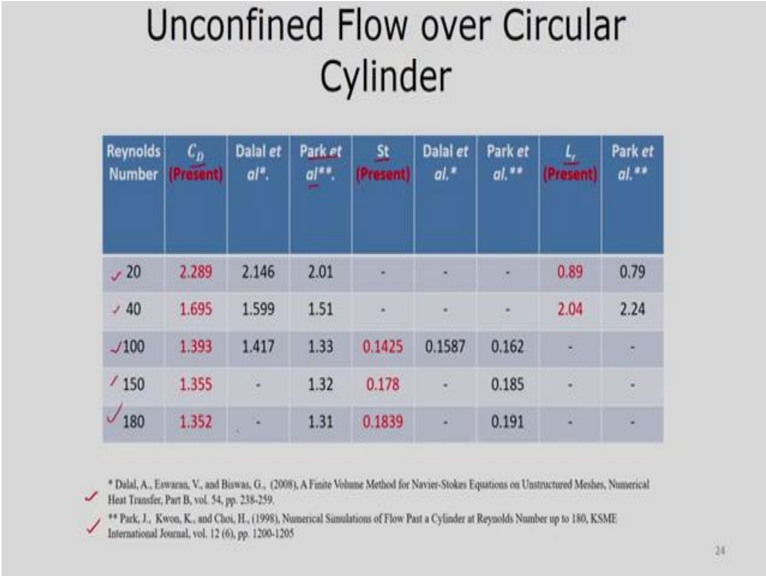
(Refer Slide Time: 40:25)



This is a problem we have taken unconfined flow over circular cylinder. So, you can see that you have a circular cylinder and you have a uniform flow inlet, these are the symmetry boundary conditions and you can have local refinement in unstructured grid, so that we have done to capture the gate interval and Reynolds number greater than 40 generally it becomes unsteady.

So this is the steady problem and this is your unsteady problem and that we have correctly captured.

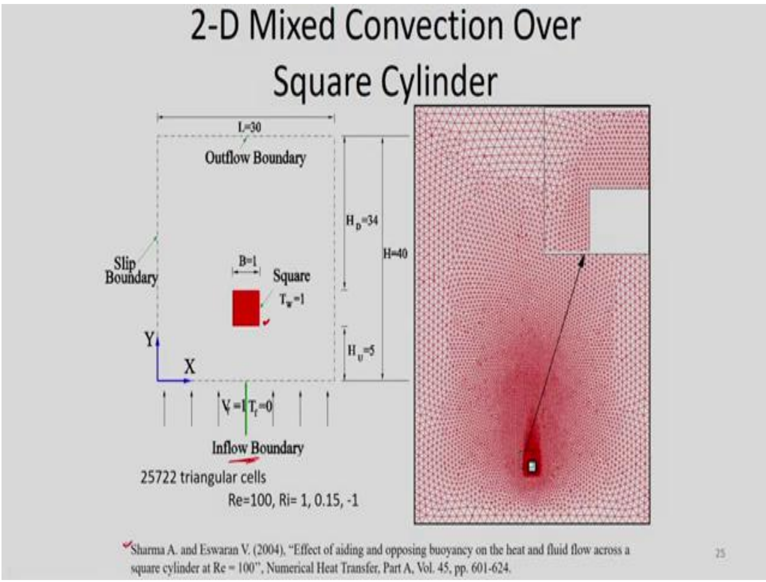
(Refer Slide Time: 40:54)



Reynolds Number	C_D (Present)	Dalal et al*.	Park et al**.	St (Present)	Dalal et al.*	Park et al.**	L (Present)	Park et al.**
20	2.289	2.146	2.01	-	-	-	0.89	0.79
40	1.695	1.599	1.51	-	-	-	2.04	2.24
100	1.393	1.417	1.33	0.1425	0.1587	0.162	-	-
150	1.355	-	1.32	0.178	-	0.185	-	-
180	1.352	-	1.31	0.1839	-	0.191	-	-

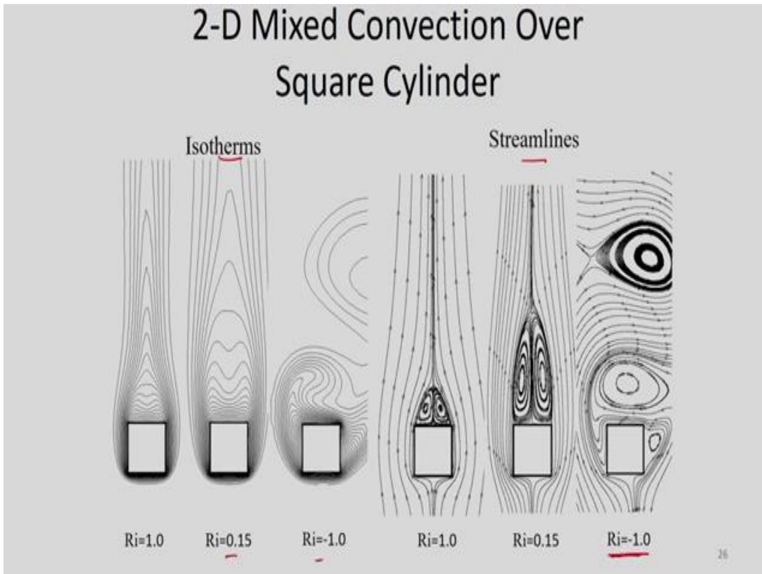
And the results of drag coefficient is 12 number and the recirculation length of steady problem we have compared with Dalal et al and Park et al. So, you can see the for different Reynolds number, we have compared these values and these comparison are very good.

(Refer Slide Time: 41:17)



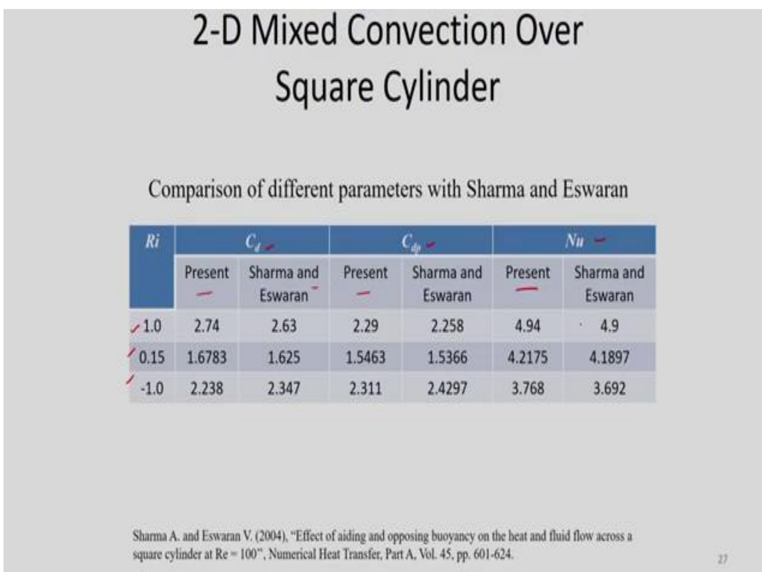
Then this is the problem of Mixed Convection where buoyancy in also included and boussinesq approximation is valid in this case and we have the square cylinder heat square cylinder, heated square cylinder, this is the inflow, so due to buoyancy there will be also there will be buoyancy effect and that we have compared our result with Sharma and Eswaran.

(Refer Slide Time: 41:47)



So, these are some isotherms and streamlines at different Richardson number you can see, so at Richardson number minus 1 you can see this is becoming unsteady.

(Refer Slide Time:42:01)

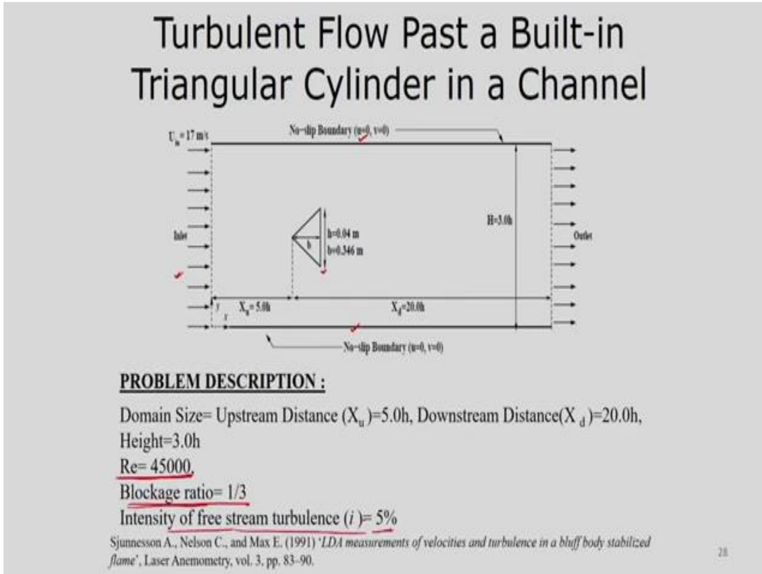


Ri	C_d		C_{dp}		*Nu*	
	Present	Sharma and Eswaran	Present	Sharma and Eswaran	Present	Sharma and Eswaran
1.0	2.74	2.63	2.29	2.258	4.94	4.9
0.15	1.6783	1.625	1.5463	1.5366	4.2175	4.1897
-1.0	2.238	2.347	2.311	2.4297	3.768	3.692

Sharma A. and Eswaran V. (2004), "Effect of aiding and opposing buoyancy on the heat and fluid flow across a square cylinder at Re = 100", Numerical Heat Transfer, Part A, Vol. 45, pp. 601-624.

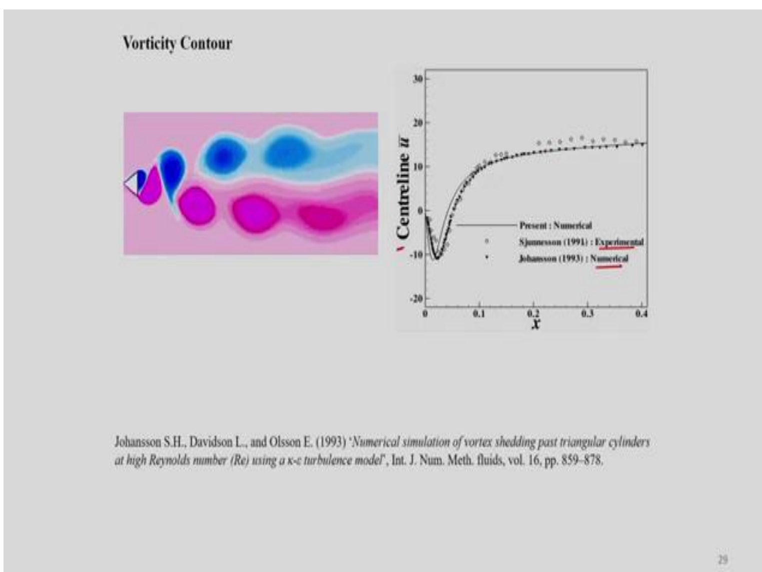
And these are some comparisons of different Richardson number, drag coefficient, pressure drag coefficient and natural number, with our these are present results and these are sum and epsilon.

(Refer Slide Time: 42:13)



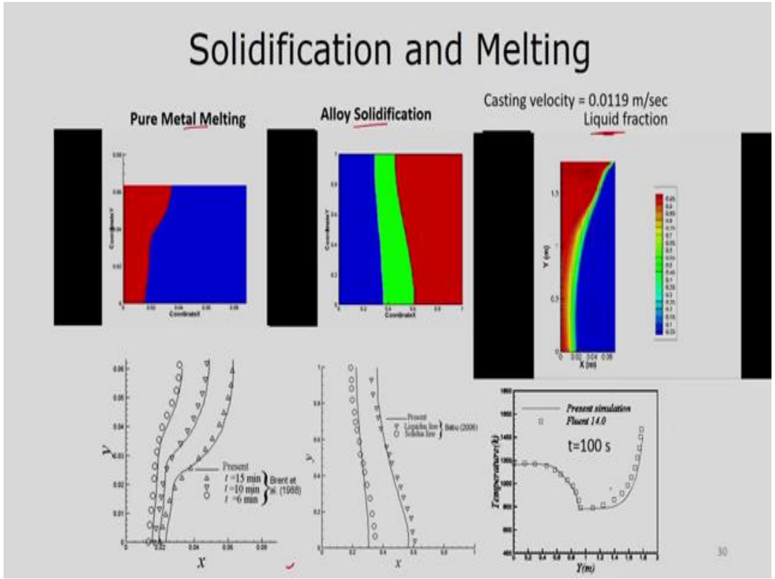
This is a turbulent flow past a built in triangular cylinder in a channel, so this is channel walls, this is your triangular cylinder and you have a inlet velocity and Reynolds number is considered 45000 so that it becomes turbulent and blockage ratio is 1 by 3 and intensity of free stream turbulence is given as 5 percent.

(Refer Slide Time: 42:34)



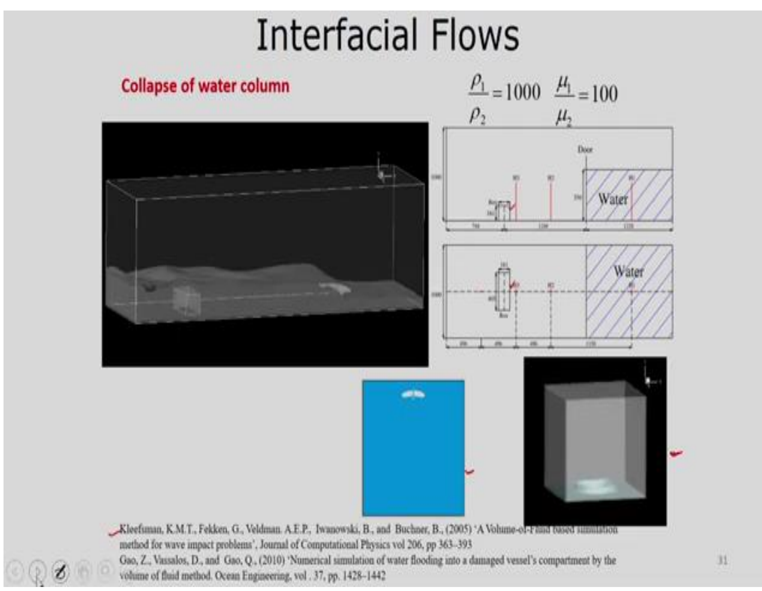
So, you can see, so these are some animations of vorticity, so flow behind this cylinder, how it looks, so periodically this are shading and these are some comparison with experimental and numerical results, so this are central line u velocity, that we have compared with the literature.

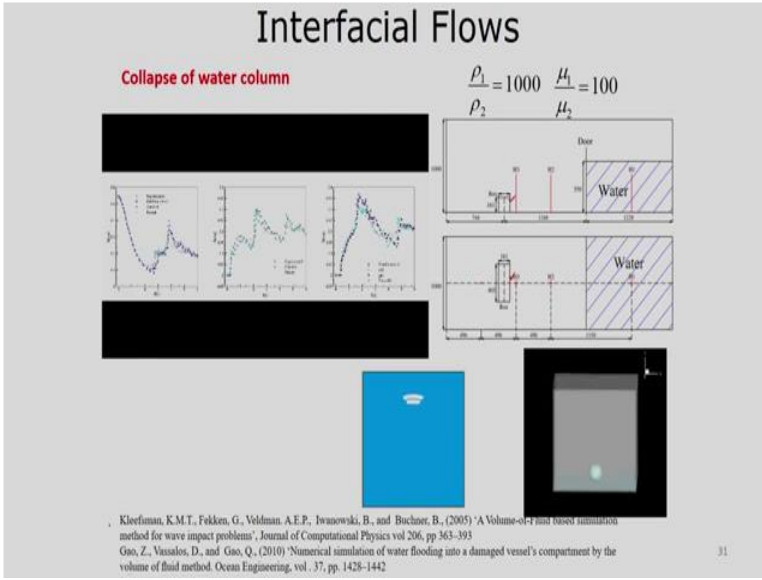
(Refer Slide Time: 42:55)



These are some phase change problem, so solidification this is the solidification problem, this is alloy solidification and this is your casting problem, so continuous casting. So, some comparisons are given of these inter-phase when it is moving with time, so that we have compared with the literature.

(Refer Slide Time: 43:19)

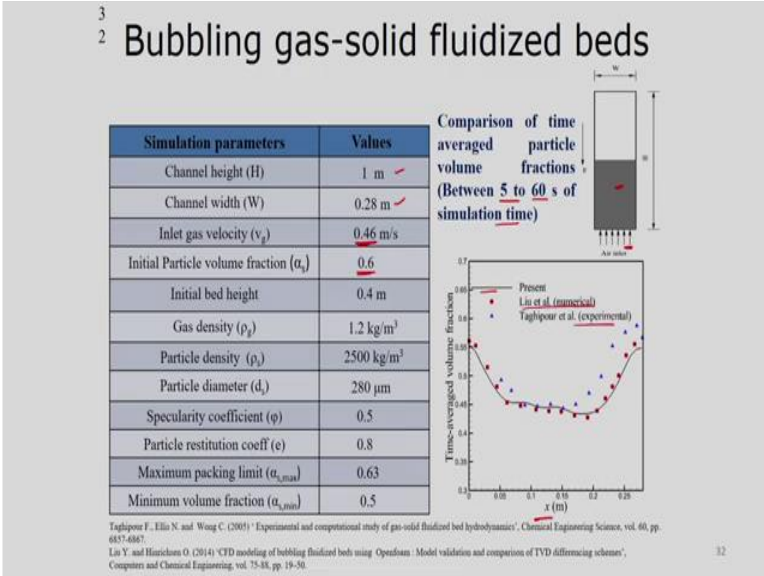




This is inter-facial flows problem, so we have one water column and there is obstruction you can see and suddenly if these inter-phase is broken then water will fall and that we have stimulated using volume of fluid method and you can see the comparison of water height at different locations with time and these results we have compare with some experimental and numerical results. So, this is you can see how the interface is moving, when the water is coming out or falling then from different angle.

So, that we have shown using volume of fluid method and these are comparisons, these are some snapshots you can see that water droplet is falling okay in thin plane then and this is the problem, 2 droplets are rising, their margin and collapsing, so these all problems we have solved using this in house solver and please remember that this are developed by the students only.

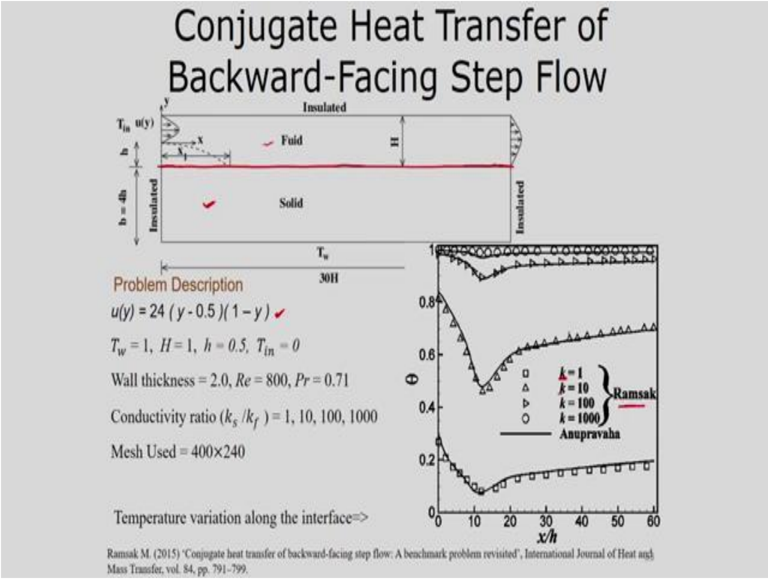
(Refer Slide Time: 44:25)



Now, this is the Bubbling gas-solid fluidized bed. So, you have solid particles here and you have some inlet velocity you can see that initial particle volume fraction is given as 0 point 6 and here initial inlet gas velocity as 0 point 46 meter per second and these are the dimensions these are some densities are given and these problem we have solved and compare the time average volume fraction, time average is done between 5 to 60 second of the stimulation time.

And you can see that our results this is our black colour solid line is our results, time volume average fraction along the a x and we have compared our result with numerical and experimental results.

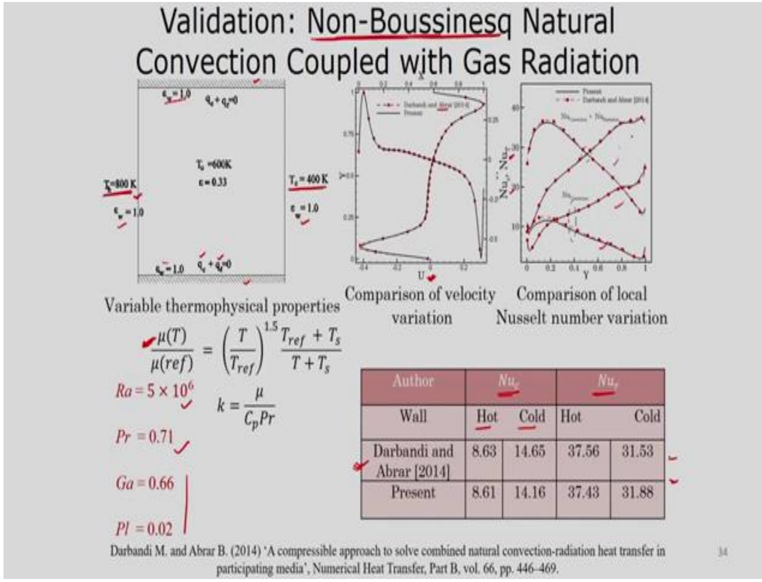
(Refer Slide Time: 45:12)



This is a problem of conjugate heat transfer, where in the solid also you solve the heat conduction equation. So, you can see this is the problem, where you have inlet is here and this is the backward facing step and this is the solid.

So and this is the fluid zone, so we have fluid and solid zone and inlet velocity you have given as parabolic and for different thermal conductivity ratio, k is the thermal conductivity ratio we define. We have solved this problems and we have potted the temperature at this interface, so this is the interface at fluid and solid right, so this is the interface from here to here, so it is the interface between fluid and solid and along this we have potted the interface temperature and compared with the literature. So you can see the it matches well.

(Refer Slide Time: 46:06)



Now, in some problem where you have a high temperature difference, like combustion, where radiation is taking place, where high temperature difference is there and radiation is taking place. So, here no longer boussinesq approximation is valid. So, in that case density may vary due to the temperature gradient, it may vary also due to the concentration gradient. So, these problem we have solved where boussinesq approximation is not valid.

So, density varies with temperature gradient and using some numerical technique we have solved this problem, where radiation is also taking place, so here we need to solve the radiative transfer equation as well along with the fluid flow and energy equation. So, you can see this is the cavity, these are the 4 walls and in the 4 walls this is Tc cold wall, 400 kelvin, this wall is 800 kelvin and emissivity are 1, so this are black surface.

Top and bottom walls are adiabatic, for radiation as well as conduction and these are also black walls and these problem we have solved, so radiation is taking place natural convection is also taking place but natural convection is taking place but boussinesq approximation is not valid, so non-boussinesq approximation we have taken.

So, here also the properties are also function of temperature. So, you can see that variable properties also we have solved, we have solved for random number 5 into 10 to the power 6, random number 0 point 71 and different other non-dimensional numbers. And these results if you see that conductive natural number, average natural number and radiative average natural number, we have calculated at hot and cold walls.

So, that we have compared with Darbandi and Abrar, so that you can see the comparison is good and here velocity we have compared along the central line, with the literature as well as the natural number, conductive natural number and total natural number N ut is the total natural number we have compare with the literature and we can see comparison is very good.

So, these results I have shown just to motivate you that these solver is written by the students, starting from scratch and they developed these general purpose CFD solver to solve a multi-physics problems. And you can see that only few results I have shown here but in last 5 years we have solved several problems and shown its accuracy just solving this validation problems.

So, to learn CFD you need to start learning programming and whatever problems in due course of time will discuss you please try to solve using some computer language may be C, C++ or Fortran. So, then actually theory you will apply to solve some fluid flow or heat transfer problems. So, in today's class first we have discussed about initial condition for marching problems you need to specify, the value at the interior domain at time t is equal to 0.

So, that is known as initial condition at the same time you need to specify the value of any variable at the boundary, so those are known as boundary conditions. So, 3 types of boundary conditions we discussed Dirichlet where value of the variable is specified then Neumann where the flux or a gradient is, normal gradient is specified at the boundary and Mixed where this combination of these two appears.

And we took some example problems and shown different types of boundary conditions. Then we discussed about the course contents and at last we have shown some results from in house solver AnuPravaha. Thank you.