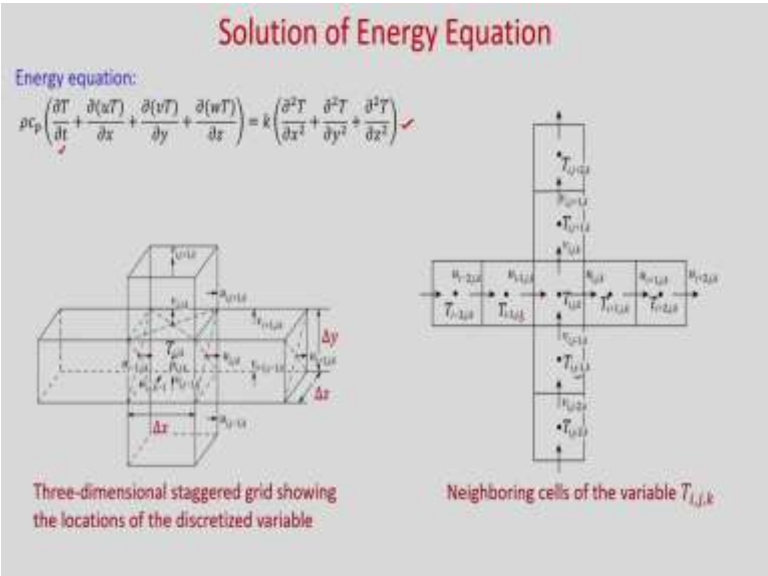
Fundamentals of Convective Heat Transfer
Prof. Amaresh Dalal
Department of Mechanical Engineering
Indian Institute of Technology, Guwahati

Module - 10
Numerical Solution of Navier-Stokes and Energy Equations
Lecture – 35
Solution of energy equation

Hello everyone. So, in last class we discretized the Navier-Stokes equations using finite difference method, also we have written the equation for pressure correction from the continuity equation. Today we will discretize the energy equation using finite difference method and then we will discuss about the boundary conditions, then we will discuss about the solution algorithm. As you know that we are using staggered grid and MAC algorithm to discretize the equations.

So, in staggered grid we solve pressure and temperature in the main cell and the velocities in staggered way. So, you can see when we will discretize the energy equation in the main cell the velocities will be available from their main control volume respectively.

(Refer Slide Time: 01:41)

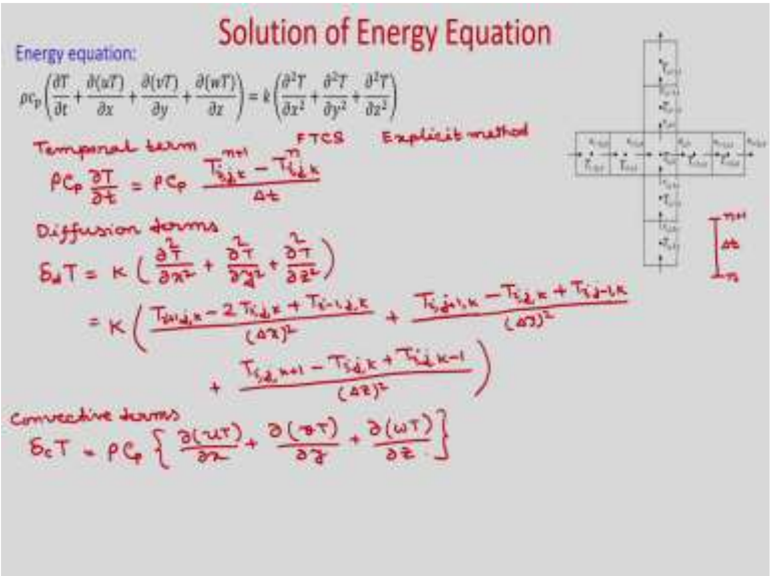


So, you can see this is the energy equation, this is the temporal term this is the convection terms and this is the diffusion terms. So, you can see temperature and pressure is solved in the main control volume.

So, when you will discretize this equation you can see that the velocities are available at this point because u we have already solved at this point and velocity v we have solved at this point and similarly for w velocity. If you see at a particular k then in two- dimension if you look then it will be easier to visualize. So, temperature we need to solve at $T_{i, j, k}$ at this main control volume and when you will use central difference you can use $\frac{T_{i+\frac{1}{2},j,k} - T_{i-1,j,k}}{\Delta x}$.

So, but the velocities at this point you can see these are available right. So, you do not need to interpolate the velocities at this point. So, this is your $u_{i,j,k}$ this is your $u_{i+1,j,k}$ and similarly for the temperature these are the neighbours $T_{i+1, j, k}$,$T_{i, j+1,k}$ $T_{i-1,j,k}$ and $T_{i, j-1,k}$ and also you will find in k direction similarly.

(Refer Slide Time: 03:13)



So, now, let us discuss the terms one by one first, let us consider the temporal term and we will use forward time central space and explicit method. So, temporal term we can discretize like this $\rho C_p \frac{\partial T}{\partial t}$. So, we are using forward time central space. So, this is explicit method and we will solve for only $T_{i,j,k}^{n+1}$ so; obviously, you know that when we are marching in time we are going from previous time n to the current time or present time at n + 1 and the time step size is Δt.

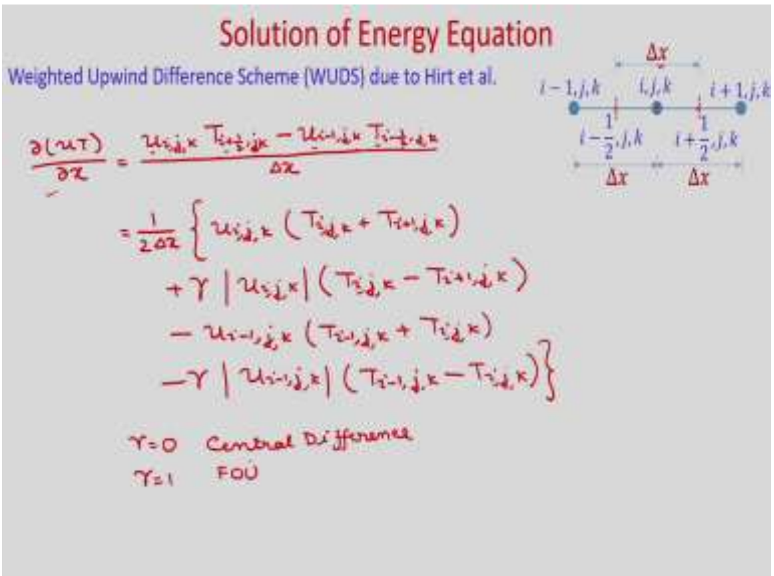
So, here you can write $\rho C_p \frac{\partial T}{\partial t} = \rho C_p \frac{T_{i,j,k}^{n+1} - T_{i,j,k}^{n}}{\Delta t}$. Now, let us consider the diffusion term, in diffusion term we have the second derivative of temperature. So, we will use central difference. So, diffusion terms. So, we can have $\delta_d T = K\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right)$. So, using central difference method if you discretize what you will get?

So, it will be,

$$\delta_d T = K\left(\frac{T_{i+1,j,k} - 2T_{i,j,k} + T_{i-1,j,k}}{(\Delta x)^2} + \frac{T_{i,j+1,k} - T_{i,j,k} + T_{i,j-1,k}}{(\Delta y)^2} + \frac{T_{i,j,k+1} - T_{i,j,k} + T_{i,j,k-1}}{(\Delta z)^2}\right).$$

So, when will you write $\delta_d T$ at time level n then all these temperature will be at time level n which is your previous time. Now when you write the convective term so, you can write $\delta_c T$ convective terms. So, $\delta_c T = \rho C_p \left\{\frac{\partial (uT)}{\partial x} + \frac{\partial (vT)}{\partial y} + \frac{\partial (wT)}{\partial z}\right\}$.

(Refer Slide Time: 06:52)



So, now let us take the term one by one and discretize using weighted upwind difference scheme. So, already we have discussed in last class in detail about this scheme, we will use to discretize the convective term in the energy equation. Now we have $\frac{\partial(uT)}{\partial x}$. So, simply you can see you can use central difference and discretize using this.

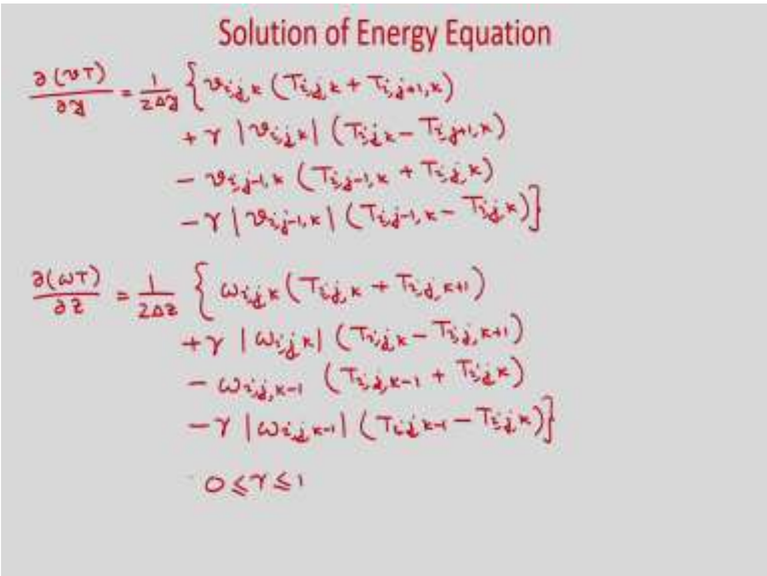
So, here you we are discretizing this term. So, we can write u. So, you can see at this point we have the value of u. So, $\frac{\partial(uT)}{\partial x} = \frac{u_{i,j,k}T_{i+\frac{1}{2},j,k} - u_{i-1,j,k}T_{i-\frac{1}{2},j,k}}{\Delta x}$ Now using this weighted upwind difference scheme you can write this.. So, if you use central difference then you will get like this, but as you are using weighted upwind difference scheme with a factor γ which varies between 0 and 1 and this is your for upwind.

So, if u > 0 it will return that positive value, if u < 0 then negative of that value then,

$$\frac{1}{2\Delta x}\begin{Bmatrix} u_{i,j,k}\left(T_{i,j,k} + T_{i+1,j,k}\right) + \gamma\left|u_{i,j,k}\right|\left(T_{i,j,k} - T_{i+1,j,k}\right) \\ -u_{i-1,j,k}\left(T_{i-1,j,k} + T_{i,j,k}\right) - \gamma\left|u_{i-1,j,k}\right|\left(T_{i-1,j,k} - T_{i,j,k}\right) \end{Bmatrix}$$

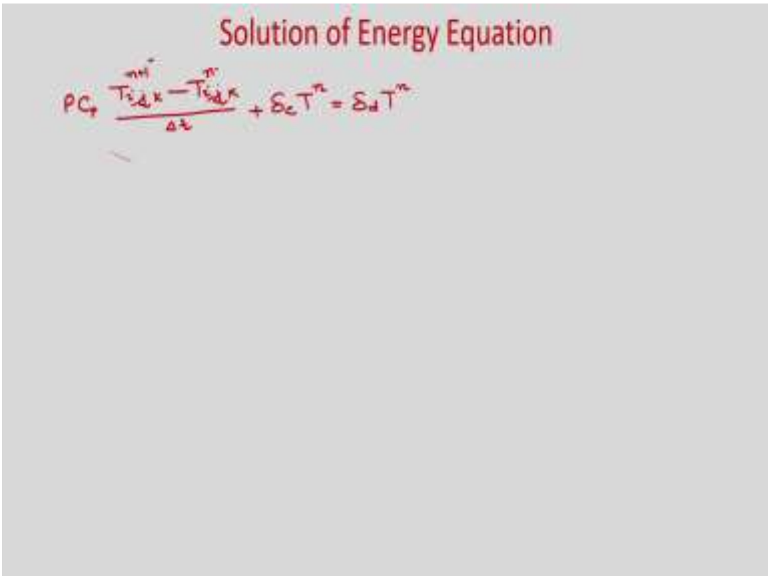
So, this is the discretization of this convective term using weighted upwind difference scheme and you know that if γ=0 then it is central difference and if γ=1 then upwind scheme.

(Refer Slide Time: 10:44)



Similarly, now let us discretize the term $\frac{\partial(vT)}{\partial y}$ and $\frac{\partial(wT)}{\partial z}$. So, you can write $\frac{\partial(vT)}{\partial y}=\frac{1}{2\Delta y}\begin{Bmatrix} v_{i,j,k}\left(T_{i,j,k}+T_{i,j+1,k}\right) \\ +\gamma\left|v_{i,j,k}\right|\left(T_{i,j,k}-T_{i,j+1,k}\right) \\ -v_{i,j-1,k}\left(T_{i,j-1,k}+T_{i,j,k}\right) \\ -\gamma\left|v_{i,j-1,k}\right|\left(T_{i,j-1,k}-T_{i,j,k}\right) \end{Bmatrix}$. Similarly you can do the discretization of $\frac{\partial(wT)}{\partial z}=\frac{1}{2\Delta z}\begin{Bmatrix} w_{i,j,k}\left(T_{i,j,k}+T_{i,j,k+1}\right) \\ +\gamma\left|w_{i,j,k}\right|\left(T_{i,j,k}-T_{i,j,k+1}\right) \\ -w_{i,j,k-1}\left(T_{i,j,k-1}+T_{i,j,k}\right) \\ -\gamma\left|w_{i,j,k-1}\right|\left(T_{i,j,k-1}-T_{i,j,k}\right) \end{Bmatrix}$, where γ varies between 0 and 1.

(Refer Slide Time: 13:17)



So, now, we have discretized the each terms. So, now, we can write the discretized form of the full energy equation temporal term we have discretized like this n + 1 and n then the convection term $\delta_c T$.

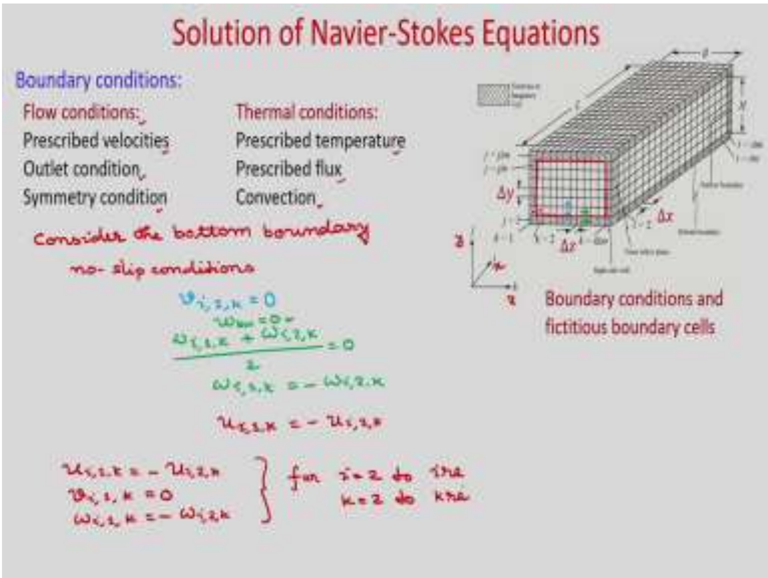
So, as you are using explicit method. So, it will be $\rho C_p \frac{T^{n+1}_{i,j,k}-T^{n}_{i,j,k}}{\Delta t}+\delta_c T^n=\delta_d T^n$. So, now, once you discretize all these terms you can put it here and you can find $T^{n+1}_{i,j,k}$

because this is the unknown all are known from the previous time level n. So, this you can find easily from this equation.

Now, if you have natural convection or heat generation in the energy equation so that will come as a source term in the governing equation. Say, if you have a buoyancy term so that is a source term you can put it in the x momentum equation, y momentum equation depending on the orientation of your problem. So, that source term you can put and you can solve the governing equation.

Similarly, in energy equation you if you have heat generation term per unit volume say q''' then you just add here q'''. So, that is that will come as a source term. So, I am not writing those equations, but easily you can put those source terms in these discretized equations. Now let us discuss about the thermal and flow boundary conditions.

(Refer Slide Time: 15:08)



So, you can see that in flow boundary conditions say at inlet we have prescribed velocities or on the solid wall we can have no slip boundary conditions means velocities will be 0. Outlet condition we can have where we can assume that fully developed condition and the gradient we can make as 0, symmetry condition also we can have.

And in thermal conditions we can have prescribed temperature or prescribed flux or we can have convection because from Newton's law of cooling at the wall your whatever heat is conducted that will be convected so that will be convection.

Now, whatever we have discussed for this geometry that i = 1, j = 1 and k = 1 all these are fictitious boundary cells. So, we are applying the boundary conditions through these fictitious boundary cells. So, whatever has cells are shown those are boundary cells and similarly i = i,i,m, j = j,i,m and k = k,i,m these are also boundary cells. So, we will use these fictitious boundary cells and we will apply the boundary conditions at the wall .

So, actually we have the boundaries here right. So, we have boundaries here. So, we have to apply the boundary conditions at wall and we will use these fictitious cells as well as your interior cells. So, we are solving the discretized equation inside this domain ok, but we need to apply the boundary conditions at the boundary fictitious cells, but at the boundary we will apply the boundary conditions and we will find the suitable values at the fictitious cells so that it will satisfy the boundary condition at the wall.

So, here you can see that i is in this direction; that means, that is your x direction, this is your y direction and this is your z direction for this geometry and if you think that it is a channel then at inlet where you have i = 1 there we can have a prescribed velocity.

And if you have a wall sidewalls then you can apply no slip boundary conditions, if you have a symmetry boundary conditions then you can apply symmetry boundary condition and at the outlet where i = i,i,m you can apply outflow boundary conditions. So, we will consider few boundary conditions flow boundary conditions and we will discuss or discretize that equations and we will find the value at the fictitious cells.

Now consider the bottom boundary consider the bottom boundary. So, if it is a bottom boundary you can see. So, j = 1 and i will vary and k will also vary. So, if you apply no slip condition, say no slip condition now what is no slip condition because the velocities will be 0 at the wall or the velocity of the fluid particle sitting on this wall will be the same will have the same velocity as the wall. So, that is your no slip conditions. So, if you see that u will be 0 at this particular point.

So, when we apply no slip condition; that means, velocities at the wall will be 0. So, now, you have u velocity, v velocity and w velocities will be 0 at the wall, but we are solving the velocities in staggered way. So, we have to find where we are solving the velocities u, v and w, from this figure you can see that your v velocity v velocity in j direction.

So, v velocity we are solving in staggered way right this is your v velocity. So, now, we can see v velocity is falling at the wall itself. And this is your v velocity in the interior points because we are solving in staggered way. So, this is your y direction. So, v velocity so, you can write $v_{i,j}$ = 1, k = 0 because v is falling at the boundary itself.

Now if you see w velocity. So, you are solving in z direction. So, this is your z direction. So, for this you can see that this is your interior point w velocity, this is your u velocity at the fictitious cell. So, it is not falling at the boundary. So, what would happen?

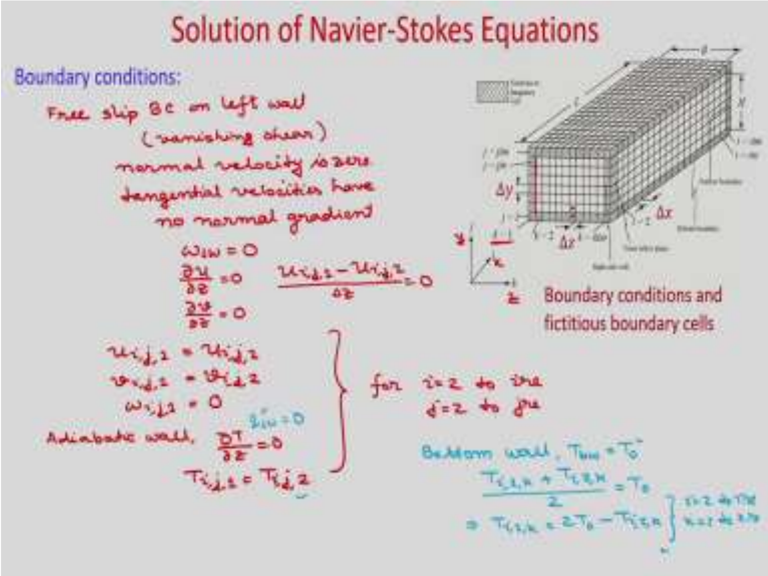
So, you have to take the average of these velocities such that the velocity at the wall will become 0 so; that means, you can write $\frac{w_{i,1,k} + w_{i,2,k}}{2} = 0$; that means, you are satisfying w at bottom wall is 0. So, at this wall you are putting the velocity 0, but you are solving w in staggered way. So, you are solving in these two points.

So, that is your average velocity you are taking 0. So, that $w_{i,1,k} = -w_{i,2,k}$. So, if you use this then essentially you will get w = 0 on the wall. Similarly, you will get for u velocity $u_{i,1,k} = -u_{i,2,k}$ because u velocity you are solving in staggered way. So, it will not fall on the wall ok. So, you have this boundary condition.

So, now from the boundary condition you can see for no slip condition on the bottom wall if you have applied then you will get $u_{i,1,k} = -u_{i,2,k}$,$v_{i,1,k}$ = 0, $w_{i,1,k} = -w_{i,2,k}$. So, these you are varying the i and k. So, for i = 2 to ire and k = 2 to kre.

Now, another boundary condition let us take let us consider the left boundary as free slip boundary condition; that means, shear stress will be 0 on this wall.

(Refer Slide Time: 22:49)



So, for free slip boundary condition on left wall . So, this is your boundary this is your boundary on the left wall now you are having on the left wall you see k = 1 and j is varying and i is varying . So, at this wall if you use the free slip boundary condition; that means, vanishing shear. There will be no shear on this wall. So, normal velocity will be 0 is 0.

So, what is normal velocity at this wall, you see in k direction. So, this is the normal velocity; that means, w will be 0 on this wall and tangential velocities have no normal gradient tangential velocities have no normal gradient.

So, you can see for velocity u and v this normal gradient will be 0; that means, you have $w_{lw} = 0$ and you have $\frac{\partial u}{\partial z} = 0$ because that is your normal gradient in z direction, this is your x, this is your y and this is your z direction. So, and you have also $\frac{\partial v}{\partial z} = 0$.

So, now, if you satisfy these conditions on this wall then you can find the velocities at the fictitious cells in terms of the interior cells. So, if you do that you will get $u_{i,j,1} = u_{i,j,2}$.

So, you can see $\frac{\partial u}{\partial z}$ what you can do. So, if you discretize it will be $\frac{u_{i,j,1} - u_{i,j,2}}{\Delta z} = 0$.

So, distance is $\Delta z = 0$. So, if it is so, now, $u_{i,j,1} = u_{i,j,2}$. Similarly, $v_{i,j,1} = v_{i,j,2}$ and $w_{i,j,1} = 0$.

And if it is adiabatic wall so, normal gradient of temperature also will be 0. So, $\frac{\partial T}{\partial z} = 0$.

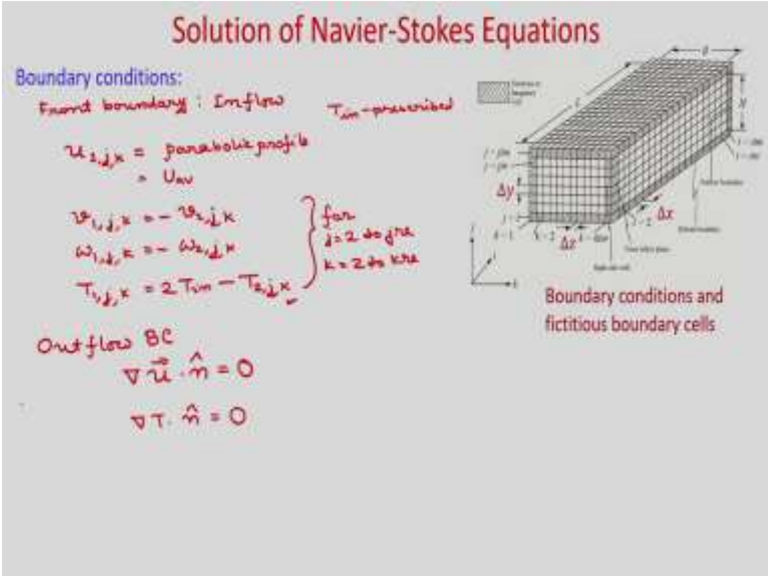
So, if it is so, then you can write $T_{i,\mathrm{j},1} = T_{i,\mathrm{j},2}$ because $\frac{\partial T}{\partial z}$ normal gradient is 0 adiabatic wall no heat flux across the wall. So, for this you can see you are varying for i = 2 to ire and j = 2 to jre and for the bottom wall if you consider that it is a constant wall temperature.

So, you can see here, you have temperature and this is your temperature. So, for a special you if you consider that bottom wall you have isothermal wall; that means, $T_{bw} = T_0$. So, now, you can see this is your interior point and this is your fictitious point, but you need to satisfy $T_{bw} = T_0$ at this point.

So, if you satisfy that then you can write, $\frac{T_{i,1,k} + T_{i,2,k}}{2} = T_0$. So, from here you can see $T_{i,1,k} = 2T_0 - T_{i,2,k}$ and this you need to vary i = 2 to ire and k = 2 to kre. So, this is for constant wall temperature boundary condition if you know the temperature on the wall T_0 and it is for adiabatic wall where q'' at left wall is 0 or left boundary.

So, it is symmetric condition also you can see that $\frac{\partial T}{\partial z} = 0$. So, this also will be valid. Now similarly for other boundary conditions also you can write say if you have an inlet. So, velocity inlet so, you can have constant velocity or you can have a parabolic profile. So, that you can write for this particular domain if at i = 1 you have a inflow condition then we can write as front boundary.

(Refer Slide Time: 28:41)



So, this is the front boundary so, you have inflow. So, if it is so, far front boundary in flow you can see it is i = 1 and j and k will vary. So, $u_{1,j,k}$ you can see $u_{1,j,k}$ will fall at the boundary itself. So, if it is so, then you can specify the parabolic profile for $u_{1,j,k}$ and if it is some constant value that also you can specify U_{av} let us say and for v velocity and w velocity it will be in staggered way.

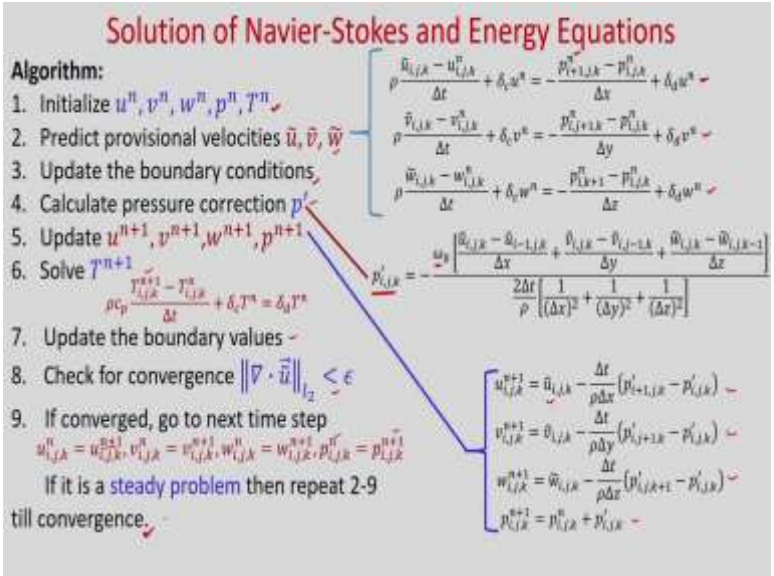
So, similarly way you can write $v_{1,j,k} = -v_{2,j,k}$ and $w_{1,j,k} = -w_{2,j,k}$ and for temperature. So, temperature if you do so, for temperature you will get $T_{1,j,k}$ let us say at the inflow your in $T_{infinity}$ is prescribed $T_{infinity}$ or let us say let us say at the inlet you have T_{in} is prescribed.

So, if it is so, you can write $T_{1,j,k} = 2T_{in} - T_{2,j,k}$ and here j will vary j = 2 to jre and k =2 to kre ok. So, prescribed temperature you have T_{in} at the inlet and you can find the T_{in} it is at the boundary. So, at the fictitious cell you will have the this temperature $T_{1,j,k} = 2T_{in} - T_{2,j,k}$ k.

Let us say that at i = iim which is your outlet. So, there you have outflow boundary condition. So, if you have outflow boundary condition. So, outflow boundary condition generally we say that normal gradients are 0. So, in general you can write n = 0 and accordingly you can write and discretize the gradient and write in terms of $T_{iim,\,j,k}$. So, you can just do as homework and for temperature also the normal gradient will be 0 ok. So, $\nabla T.\hat{n} = 0$ at the outflow boundary condition.

So, now we have the discretized equations for Navier-Stokes equations, we have pressure correction equations we have pressure correction equation and also we have discretized energy equation. So, now, let us discuss about the solution algorithm. So, we have already discretized these Navier-Stokes equations.

(Refer Slide Time: 32:13)



So, first you initialize the velocities at T = 0 and or previous time level you can find the values from the solution and first we initialize the values at T =0 then you predict the provisional velocities, you have seen that in the discretized equation the pressure at n +1 level is not available. So, we have used the previous pressure value previous time level pressure value and we have found the provisional velocities from these equations.

So, these are the equations taking the pressure value from the time level n ok. So, u tilde, v tilde and w tilde you find, then update the boundary conditions then you calculate the pressure equation. So, from this expression already we have derived from the continuity equation. So, you can see this is the equation. So, $P'_{i,j,k}$ you can find, omega naught is your overall relaxation factor and; obviously, this is the $\nabla, \vec{u}$.

So, this equation you can use to find P', once you know the P' and the provisional velocities you can update the velocities and pressure from these equations. So, you can see u tilde and the pressure corrections are known. So, you can use these equations to update the velocities and pressure you can update as $P^{n+1} = P^n + P'$, once you calculate this then you solve for the temperature because you can see in the temperature velocity is adequate.

So, these are the velocities at that time level. So, you can use those velocities and solve this equation to find T^{n+1}. Then again you update the boundary values then check for convergence, for the velocities you can check as well as for energy equation also similarly you can check, then if converged then go to the next time step. So, when you will go to the next time step you put these n plus 1 values at u n so that you can use these values.

So, all u v w your and for pressure you are taking from the p n value you are putting P^{n+1}. So, that it will be your initial in the initial values in the next time step. So, this way you will continue for the unsteady problem, but if it is a steady problem then repeat 2 to 9 till convergence.

For steady problem you can use a unsteady solver and you can merge in a pseudo tangent way and till convergence you can repeat 2 to 8 and you can find the converged solution for velocities and temperature, but if it is an unsteady problem then; obviously, you see that it will you have to go to the next time step and it depends on you that how much time you want to march.

If you have mixed convection problem or natural convection problem then; obviously, you can see temperature and velocities are coupled because when you will solve the momentum equations you have the temperature there and anyway in that energy equation you have the velocities so; obviously, you have to solve in a coupled way.

So, that is why you are solving the temperature after just after the velocity solution in each time step, but if it is only forced convection where both coupling is not there; that means, in when you solve the momentum equations you do not need the temperature then you first converged for a steady state problem, first you converge the velocities then separately you just solve energy equation. So, that is known as segregated method.

For forced convection problem you can use segregated method because after solution of the velocities separately you can solve the energy equation, but if it is buoyancy driven flow means natural or mixed convection then; obviously, you need to solve the energy equation along with the velocities because these are both way coupled.

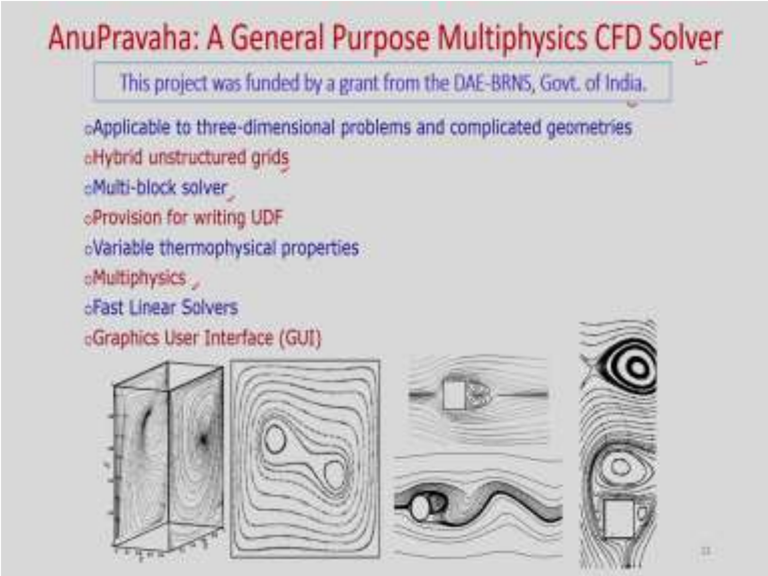
So, we have discretized the Navier-Stokes equations and energy equations using finite difference method and we have used MAC algorithm to solve these equations you can solve these equations numerically when you cannot have the analytical or exact solutions of the governing equations.

So, you see that if you consider a channel flow, in channel flow when it is developing both thermal and hydrodynamic boundary layers are developing then; obviously, it is very difficult to study analytically, but you can use numerical techniques and find the velocity distribution as well as you can find the heat transfer parameters like heat transfer coefficient and Nusselt number easily.

You can find local Nusselt number as well as average Nusselt number in the developing region and if you have a complicated geometry or where you cannot have a simplified form of the governing equations then you cannot have the analytical solutions. So, it is more convenient to use numerical techniques for these kind of problems.

Now, I will show some solutions of heat transfer problems which are solved using CFD using numerical techniques; however, the results whatever I will be showing we have used in house solver AnuPravaha which is actually finite volume based solver.

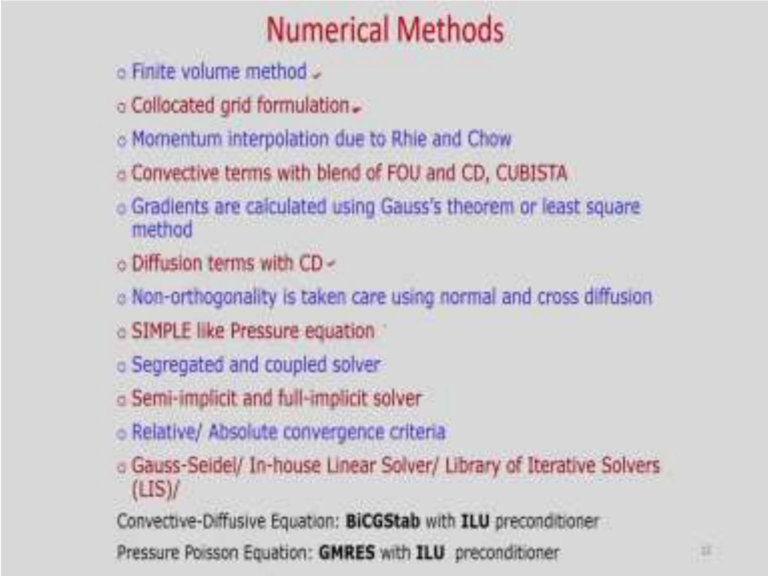
(Refer Slide Time: 38:38)



So, you can see AnuPravaha we have this in house solver which is a general purpose multi physics CFD solver and this project was funded by a grant from the DAE- BRNS, Government of India and we have these features it is applicable to three- dimensional problems and complicated geometry we have used unstructured grids, we have multi block solvers and it is multi physics ok. We can solve different kinds of problem.

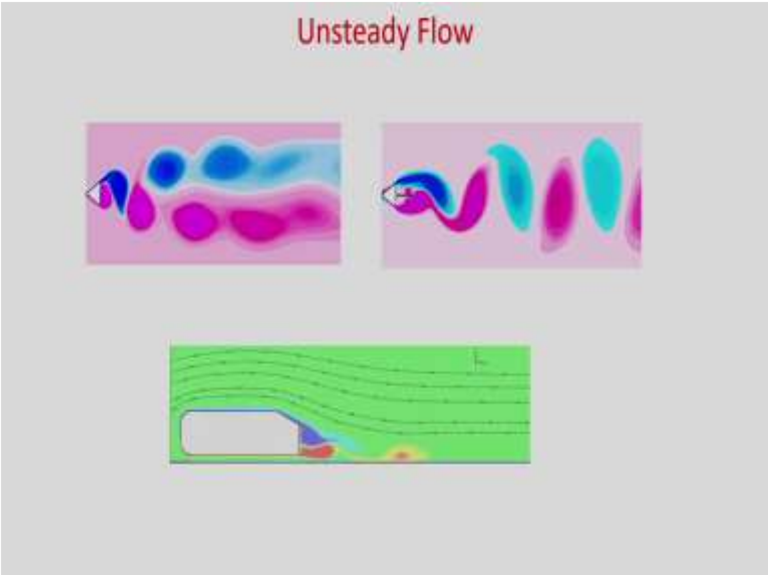
So, you can see these are the some results of natural convection and lid driven cavity or unsteady flow, you can see flow over square cylinder, circular cylinder, mixed convection over a square cylinder.

(Refer Slide Time: 39:20)



So, we have used actually finite volume method here and collocated grid formulation we have used and as collocated grid you know that there is a problem of velocity and pressure decoupling and we have used momentum interpolation, and diffusion terms with central difference we have discretized.

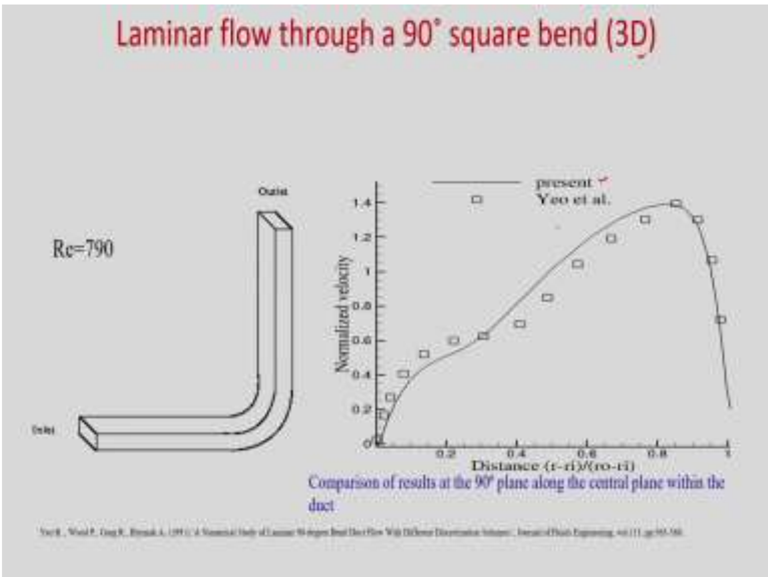
(Refer Slide Time: 39:41)



And we have used simple algorithm simple like algorithm. So, you can see these are some unsteady flows these simulations are carried out using AnuPravaha solver and you can see the animation. So, flow over a triangular cylinder, this is flow over a triangular cylinder with a splitter plate. So, here you can see one splitter plate is there and how the von Karman vortex streets are formed behind the cylinder and it is seated behind the cylinder.

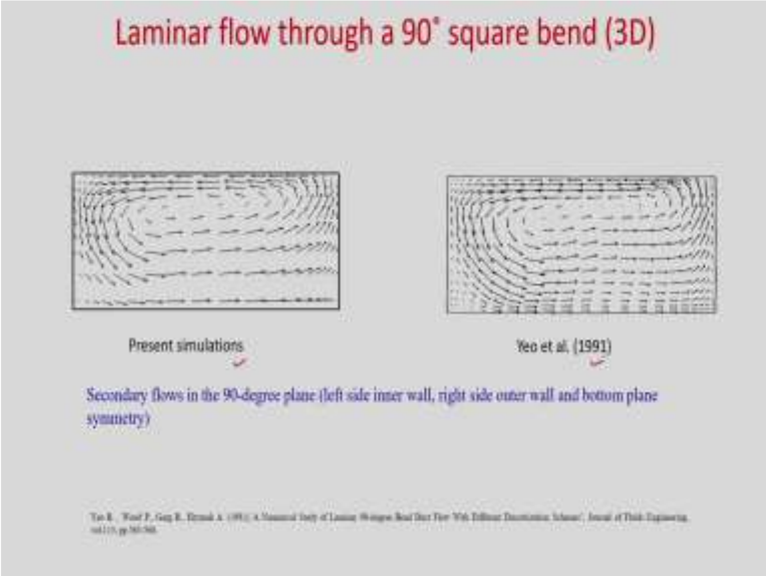
So, you can see these are vorticity contours and how it is transporting behind the cylinder and this is your kind of armoured body problem. So, behind this body it is moving in the negative x direction so; obviously, you can see the flow physics behind this body and interacting with the wall.

(Refer Slide Time: 40:27)



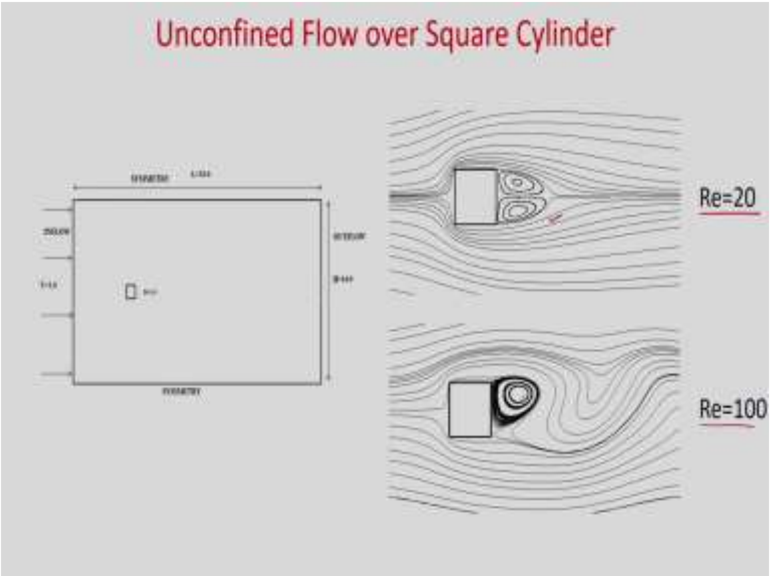
So, this is your wall. So, you can see this is laminar flow through a 90^0 square bend. So, this is your inlet, this is your outlet, you have Reynolds number 790. So, normalized velocities in distance $\frac{(r-r_0)}{(r_0-r_i)}$ this is your solid line is our result and these square is your Yeo et. al.

(Refer Slide Time: 40:55)



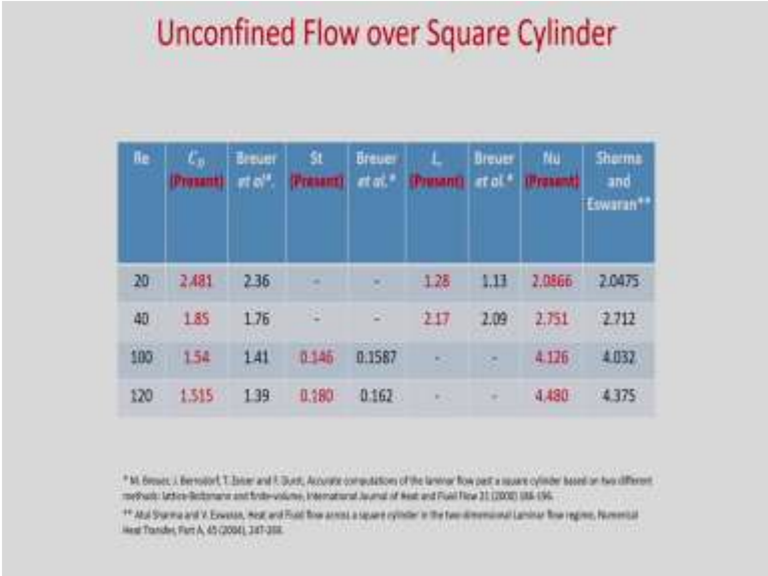
And if you see the flow physics at 90^0 plane left side inner wall right side outer wall and bottom plane is symmetry.

(Refer Slide Time: 41:09)



So, you can see how the flow physics is looking. So, this is your present simulation and this is your Yeo et al simulation. Now you consider the convective flow over a square cylinder and you know that after a certain Reynolds number it will become unsteady flow. So, you can see this is Reynolds number 20, this is your symmetry vertices are formed this is in steady region and for Reynolds number 100 this is an unsteady region.

(Refer Slide Time: 41:32)


Unconfined Flow over Square Cylinder

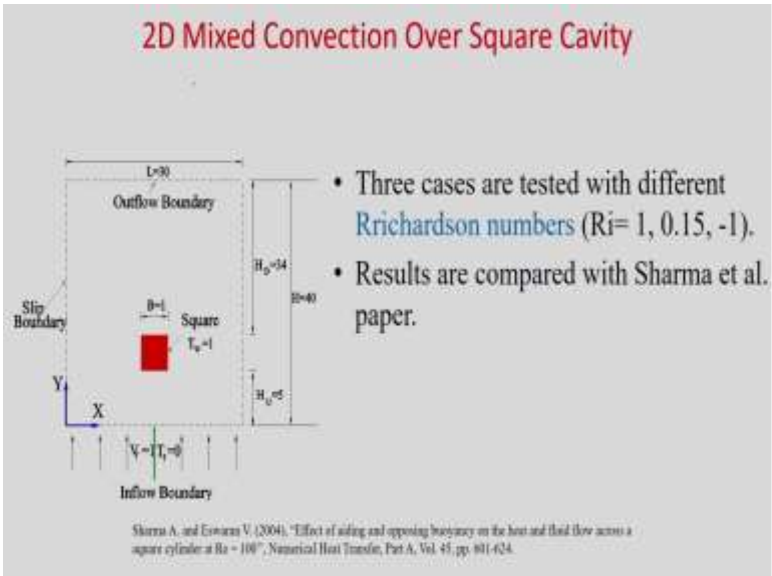
Re	C_D (Present)	Breuer et al*.	St (Present)	Breuer et al.*	L (Present)	Breuer et al.*	Nu (Present)	Sharma and Eswaran**
20	2.481	2.36	-	-	1.28	1.13	2.0866	2.0475
40	1.85	1.76	-	-	2.17	2.09	2.751	2.712
100	1.54	1.41	0.146	0.1587	-	-	4.126	4.032
120	1.515	1.39	0.180	0.162	-	-	4.480	4.375

* M. Breuer, J. Bernsdorf, T. Zeiser and F. Durst, Accurate computations of the laminar flow past a square cylinder based on two different methods: lattice-Boltzmann and finite-volume, International Journal of Heat and Fluid Flow 21 (2000) 186-196.

** Atul Sharma and V. Eswaran, Heat and Fluid flow across a square cylinder in the two-dimensional laminar flow regime, Numerical Heat Transfer, Part A, 45 (2004), 247-269.

And these are some drag coefficients 12 number for the unsteady flows and the Nusselt number on the cylinder you can see. So, we have calculated for different Reynolds number 20 and 40 are steady flows and Reynolds number 100, 120 unsteady flows. So, and these are also compared with some literature. Now, you can see 2-D mixed convection over a squared cylinder.

(Refer Slide Time: 41:52)



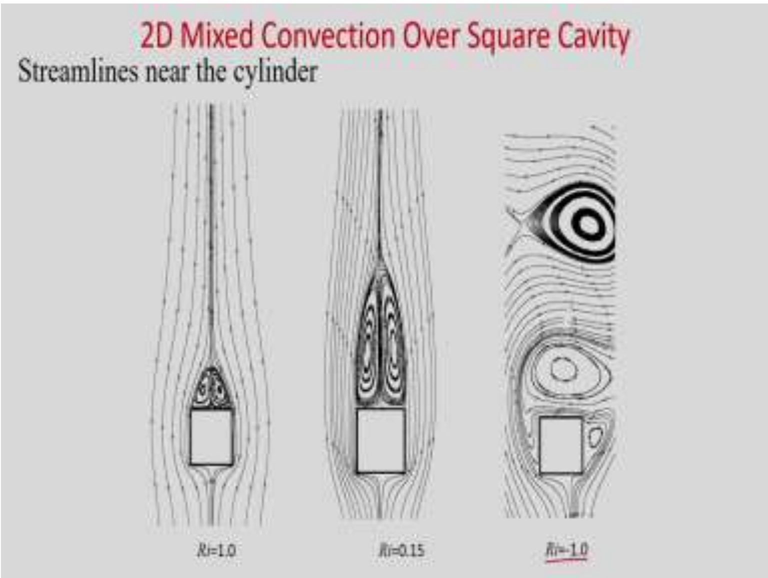
So, we have a mixed convection you have already studied it, for Richardson number 1, 0.15 and -1 these are simulated for Reynolds number 100.

(Refer Slide Time: 42:06)

So, you can see the isotherms near the cylinder. So, Ri = 1, Ri= 0.15, Ri=- 1.

So, Ri=- 1you can see it becomes unsteady.

(Refer Slide Time: 42:19)



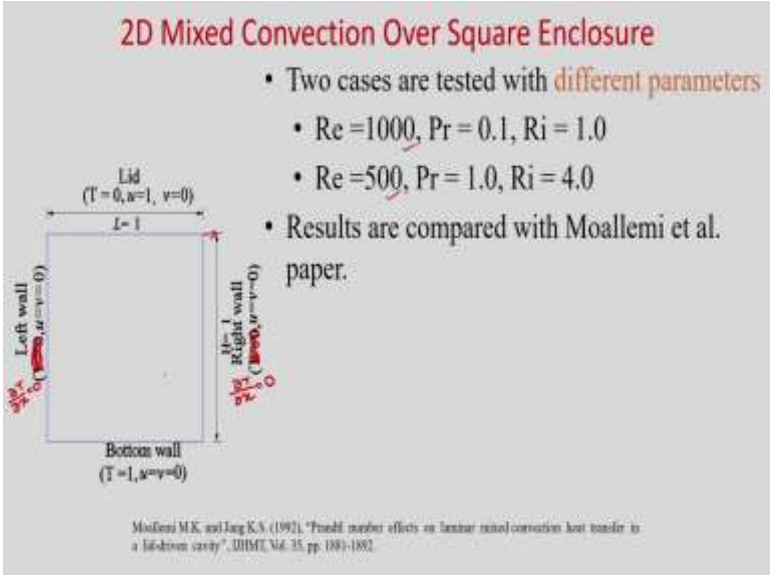
These are streamlines and from streamline plot you can see for Ri= - 1these vortices are seated behind the square cylinder.

(Refer Slide Time: 42:29)

Ri	C_d		C_{d}		Nu at the square	
	Present	Sharma et al.	Present	Sharma et al.	Present	Sharma et al.
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

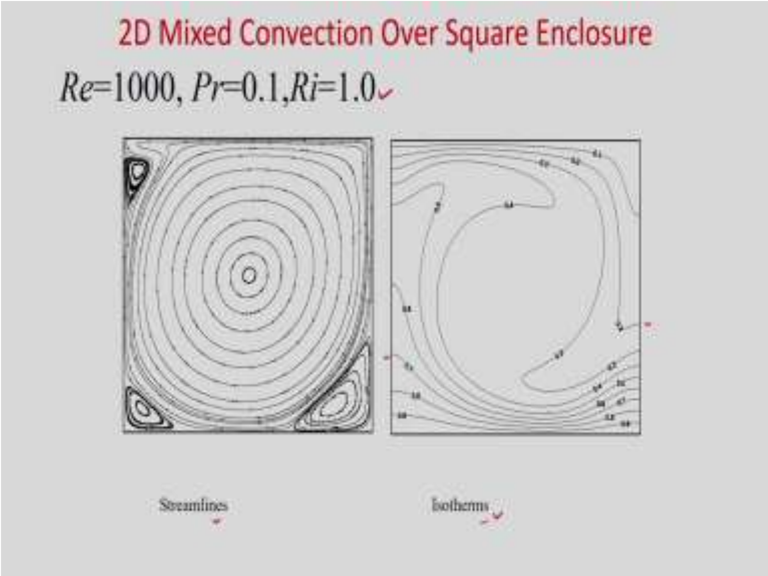
These are some comparison of drag coefficient with the literature and the Nusselt number at the square cylinder.

(Refer Slide Time: 42:38)



Then, we have 2-D mixed convection over a square enclosure. So, you can see this is your cavity the upper lid is moving in the x direction and temperature is T_0 and bottom wall is T_1 and these are also maintained at T_0 sidewalls. So, Reynolds number 1000 and 500 for different Prandtl number, Richardson number these are simulated.

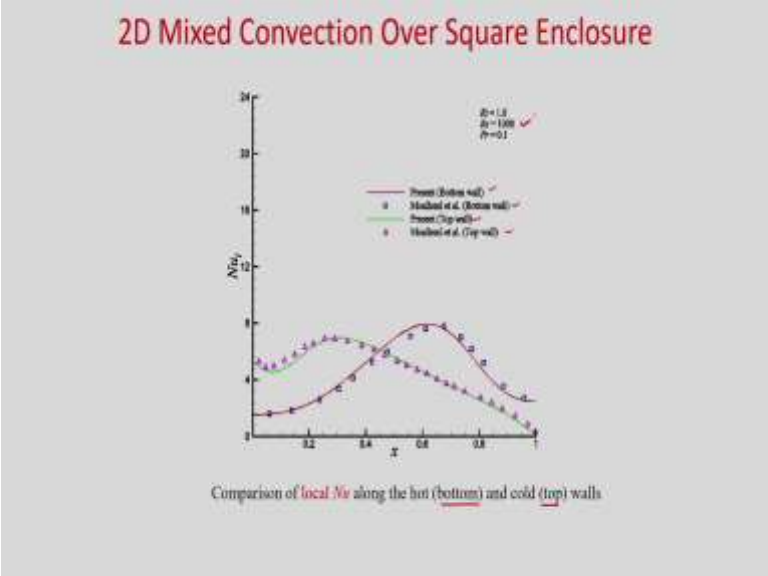
(Refer Slide Time: 43:08)



And you can see for these non-dimensional numbers, this is the streamlines and this is the isotherms. So, here the at the sidewalls you have the temperature not T = 0 this is your adiabatic wall. So, $\frac{\partial T}{\partial x} = 0$ and this right wall also you have adiabatic wall. So, $\frac{\partial T}{\partial x} = 0$.

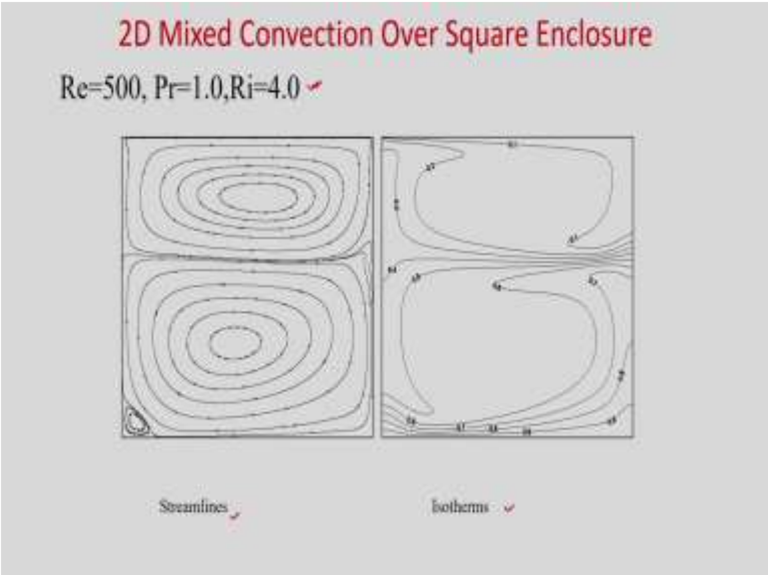
Now, for these boundary conditions you can see the stream lines for this non-dimensional numbers. So, in the stream lines you can see there are vortices near to the corners and these are the isotherms and you can see sidewalls are adiabatic. So, temperature contour is cutting this walls normally then you will get because $\frac{\partial T}{\partial x} = 0$. So, you can see from the simulations and top wall is actually 0 and bottom wall is 1. So, we can see how the isotherms are varying.

(Refer Slide Time: 44:06)



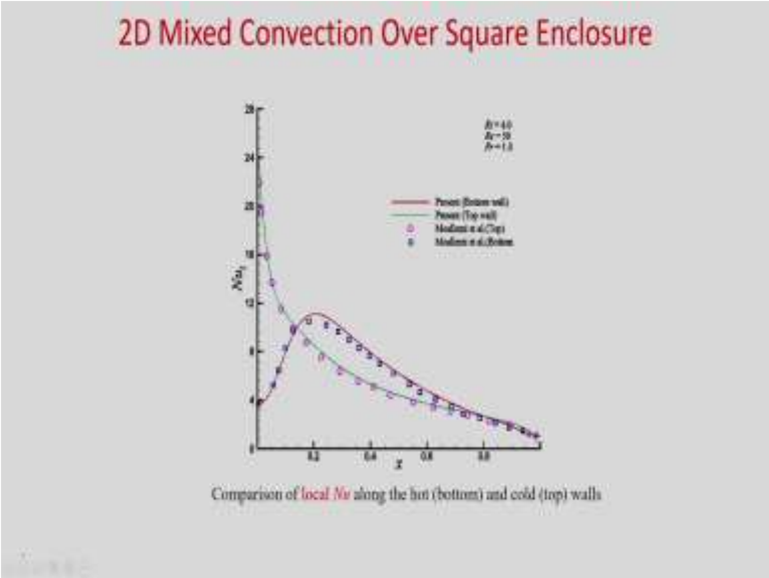
And this is the local Nusselt number, on the bottom and top wall. So, you can see red colour is your bottom wall this is the present simulation from the literature and top wall is this green colour and this is your from literature, you can see there is a good match for these non- dimensional numbers.

(Refer Slide Time: 44:26)



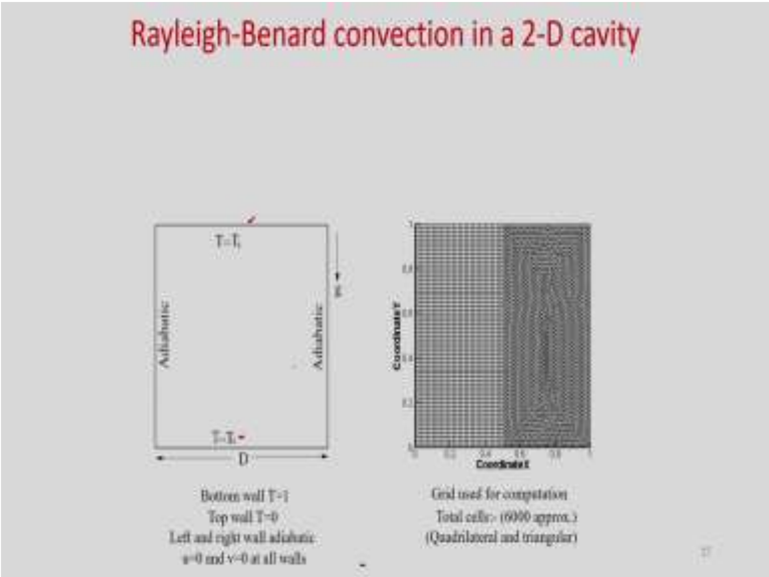
So, this is for another set of parameters this is a streamlines and this is the isotherms.

(Refer Slide Time: 44:34)



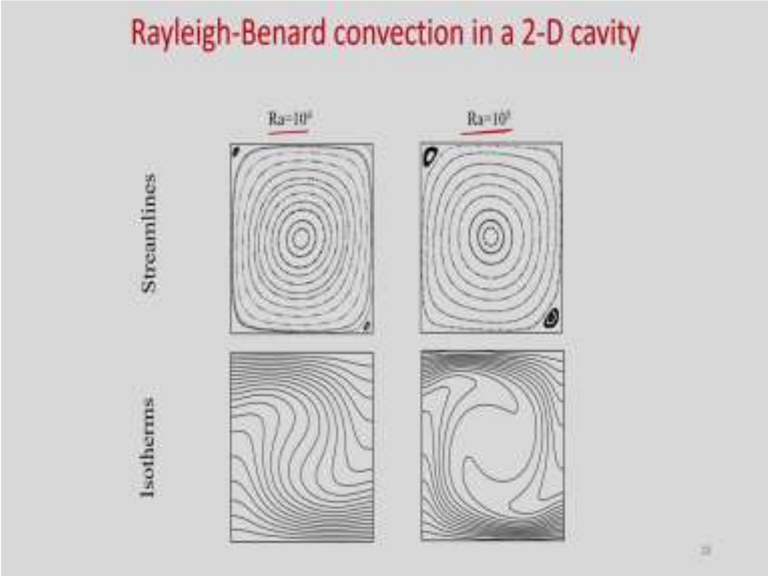
And these are the local Nusselt number variation. Then, we have a Rayleigh-Benard condition in 2-dimensional cavity.

(Refer Slide Time: 44:37)



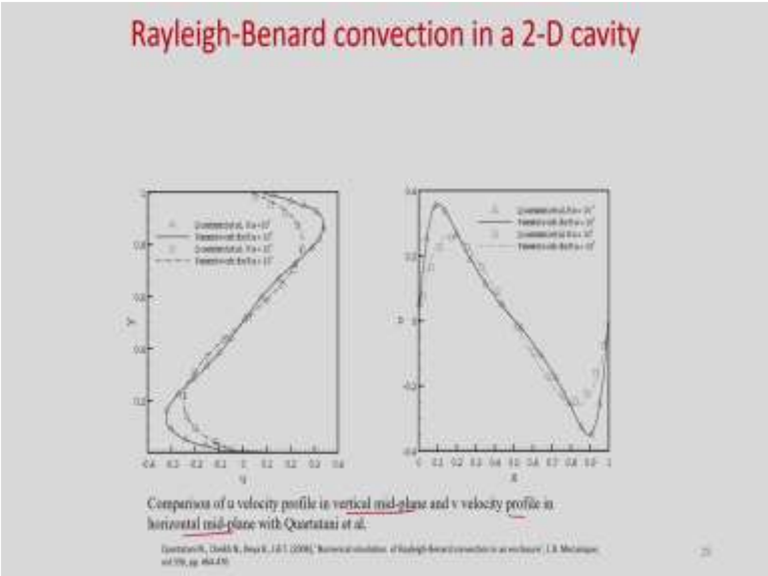
So, Rayleigh-Benard convection you know that bottom wall is maintained at T_h , this is your higher temperature than the top wall temperature and sidewalls are adiabatic.

(Refer Slide Time: 44:54)



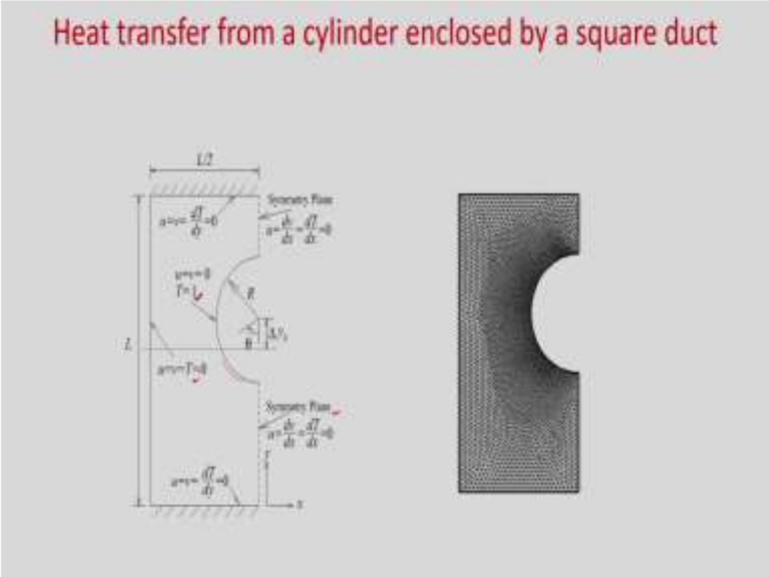
And these are the streamlines and isotherms for different Rayleigh number 10^4 and 10^5. So, right walls are adiabatic. So, the isotherms are cutting the wall normally.

(Refer Slide Time: 45:07)



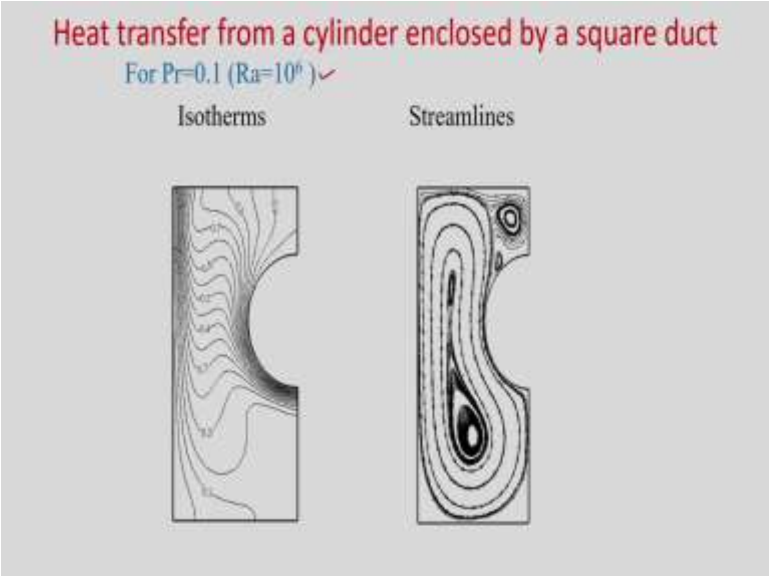
So, these are some velocities u and v. So, u velocity profile in vertical mid-plane and y velocity profile in horizontal mid-plane compared with the literature.

(Refer Slide Time: 45:19)



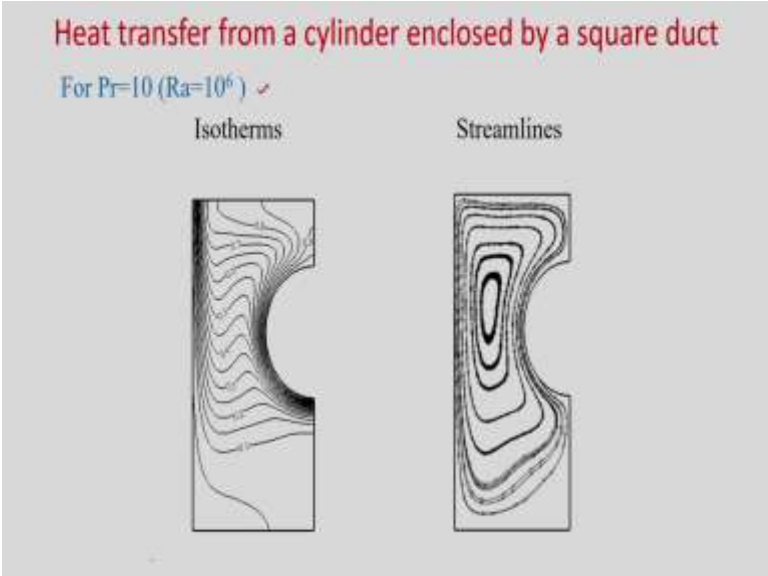
This is a heat transfer from a cylinder enclosed by a square. So, half of the geometry is shown here because due to symmetry we have solved only half of the domain and this is your symmetry plane and top and bottom walls are adiabatic, this cylinder circular cylinder is maintained at hot temperature T_1 and sidewall is maintained at temperature T_0.

(Refer Slide Time: 45:43)



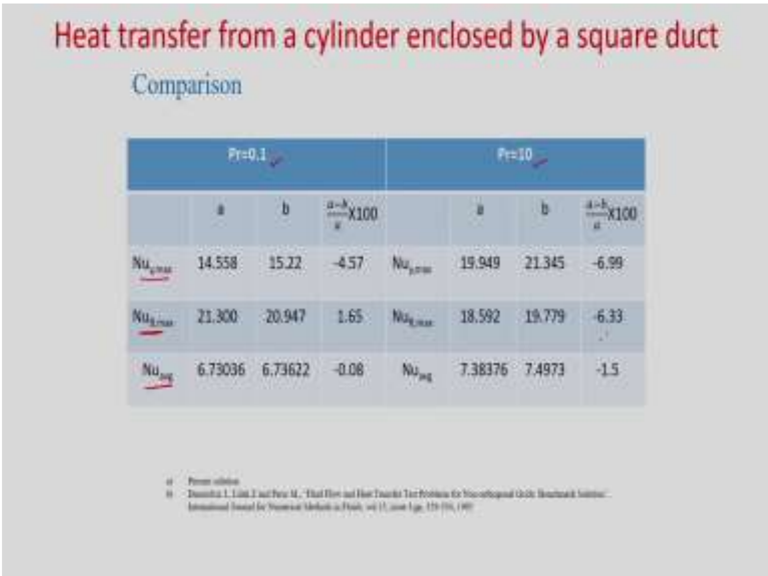
So, you can see for this setup non dimensional parameters, this is the isotherms and these are the streamlines.

(Refer Slide Time: 45:50)



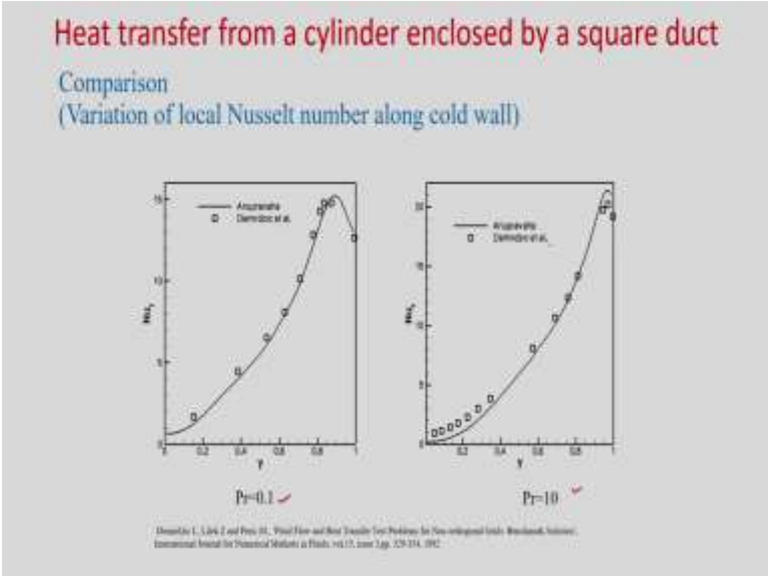
And for Pr = 10 and Ra= 10^6 these are the isotherms and streamlines.

(Refer Slide Time: 45:58)



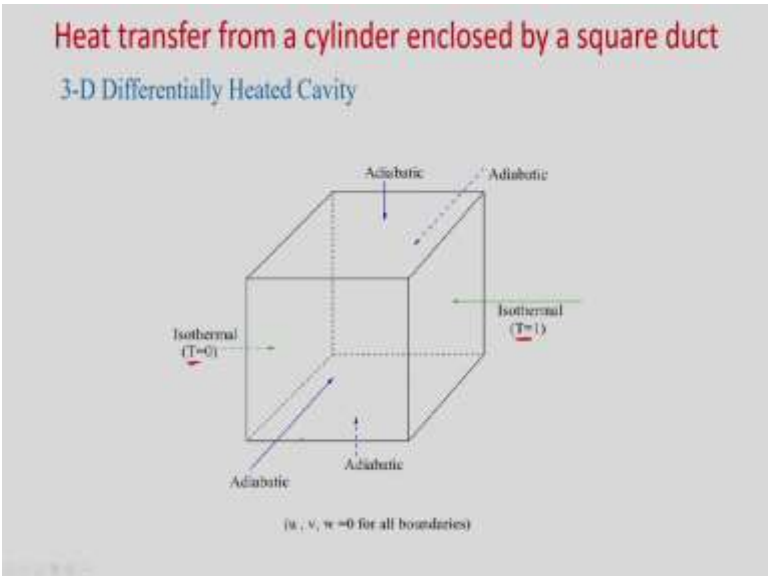
And this is some comparison, local average Nusselt number and at sidewall what is the maximum Nusselt number and on the cylinder what is the maximum Nusselt number. So, a is our result and b is from this literature and the percentage difference in the result you can see for two different Prandtl numbers.

(Refer Slide Time: 46:21)

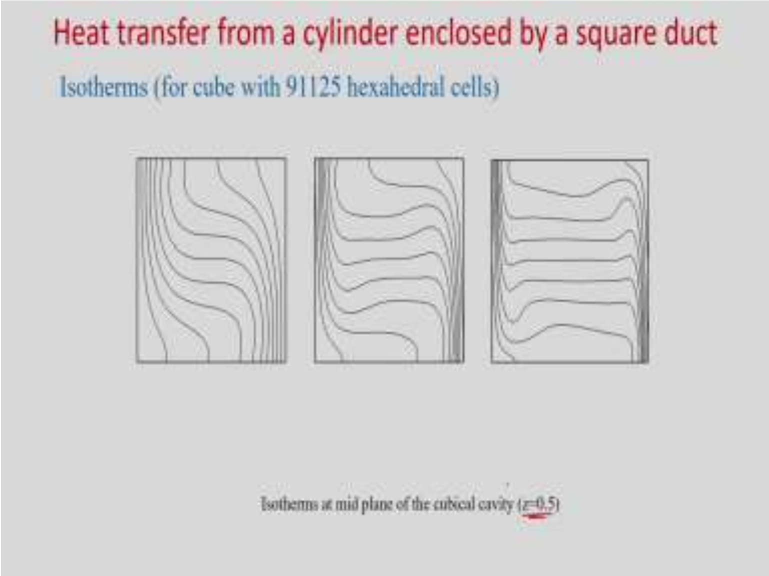


So, this is some local Nusselt number variation along the cold wall for two different Prandtl numbers and it matches well with the Demirci et al. These differentially heated cavity so, now, 3-D problem. So, these two walls are differentially heated you can see this is your T = 1 and T = 0 and other 4 walls are adiabatic.

(Refer Slide Time: 46:29)

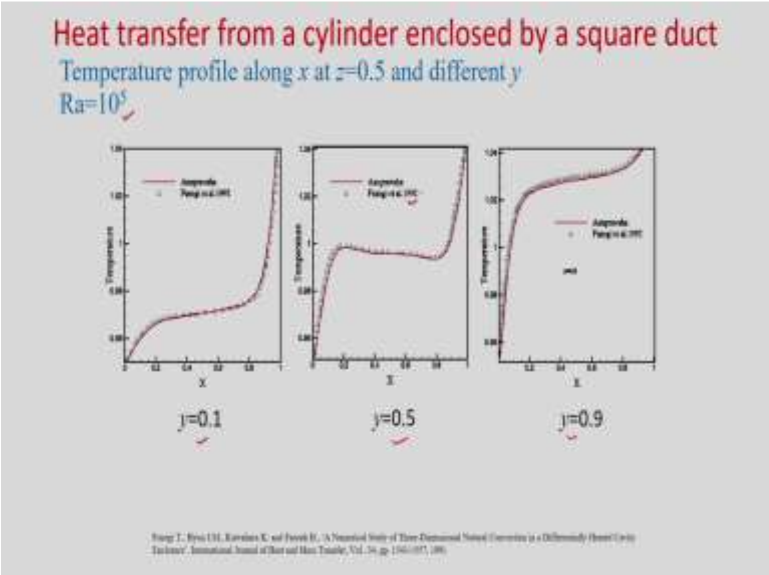


(Refer Slide Time: 46:42)



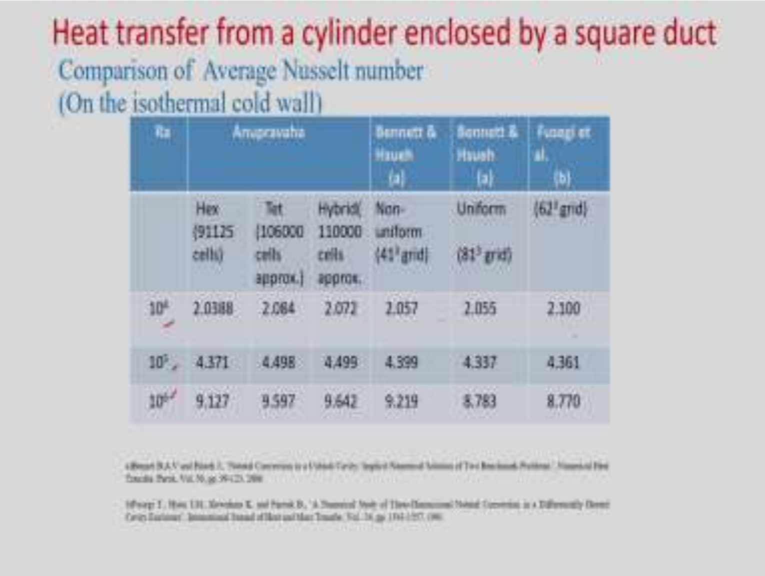
So, for this you can see the isotherms at mid-plane of the cubical cavity at z = 0.5.

(Refer Slide Time: 46:48)



And temperature profile along x at z = 0.5 at different y Ra= 10^5. So, you can see y =0.1, 0.5, 0.9 how the temperature is varying along x with the and this results are compared with Fusegi et al.

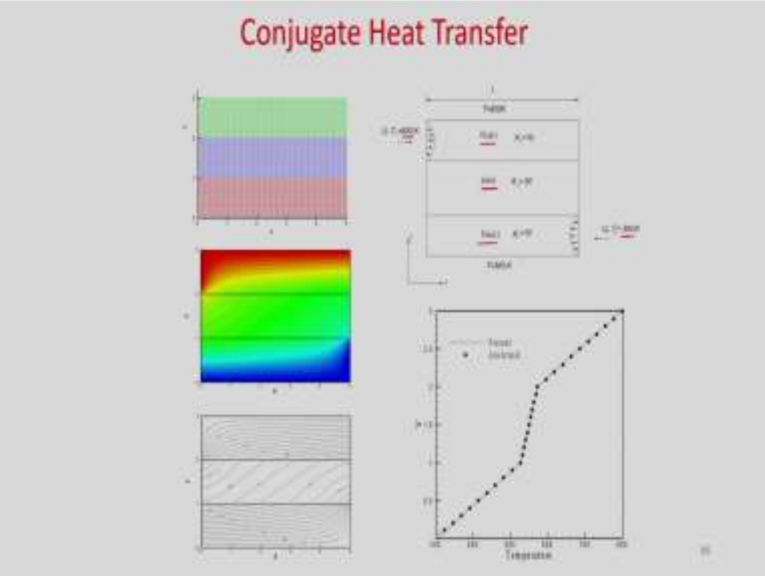
(Refer Slide Time: 47:10)



Ra	Anupravaha			Bennett & Hsueh (a)	Bennett & Hsueh (a)	Fusegi et al. (b)
	Hex (91125 cells)	Tet (106000 cells approx.)	Hybrid(110000 cells approx.	Non-uniform (41^3 grid)	Uniform (81^3 grid)	(62^3 grid)
10^4	2.0388	2.084	2.072	2.057	2.055	2.100
10^5	4.371	4.498	4.499	4.399	4.337	4.361
10^6	9.127	9.597	9.642	9.219	8.783	8.770

These are some comparison of average Nusselt number at different Rayleigh number. So, this is our solution using different types of grid and these are from literature and you can see there is a good match.

(Refer Slide Time: 47:26)



Also we can solve conjugate heat transfer, what is conjugate heat transfer? In conjugate heat transfer we solve the heat conduction equation in the solid domain and in the fluid domain we solve the fluid flow equations and the energy equation. So, like in heat exchanger. So, parallel or cross fluid heat exchanger you can see that it will be separated by solid walls. And in the solid walls we calculate the heat transfer.

So, you can see this is your solid in the middle and this is your fluid domain fluid is flowing from left to right and this is the fluid two domain where fluid flow is taking place from right to left and temperature are maintained at T = 300 K on the bottom 800 K at the top and this is the inlet temperature here and this is the inlet temperature here.

So, you can see how the isotherms are varying, in the solid also you can see how the isotherms are varying. So, here you can see. So, this is your solid and these are the fluid domains and you can see the temperature along y so how it is varying.

In today's class first we solved the energy equation using finite difference method, then we have discussed about the thermal and flow boundary conditions. We have used fictitious cell method and found the velocities and the temperature at the fictitious cell. Then we discussed the solution algorithm where you are solving Navier-Stokes equations as well as the energy equation. Then we have shown some heat transfer results use for the problems solving from the in house solver AnuPravaha.

So, we have shown for forced convection and mixed convection, results in terms of isotherms, streamlines as well as local Nusselt number and average Nusselt number. So, you can use these numerical simulations for the problems where you cannot have the analytical solutions and if you have a complicated geometry also you can use numerical technique because you cannot have the analytical solution available.

Thank you.