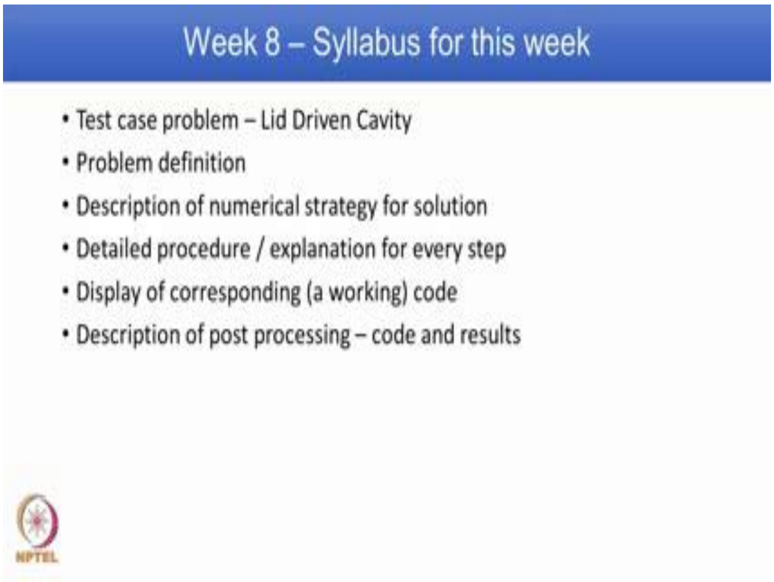
Foundation of Computational Fluid Dynamics
Dr. S. Vengadesan
Department of Applied Mechanics
Indian Institute of Technology, Madras

Lecture - 38

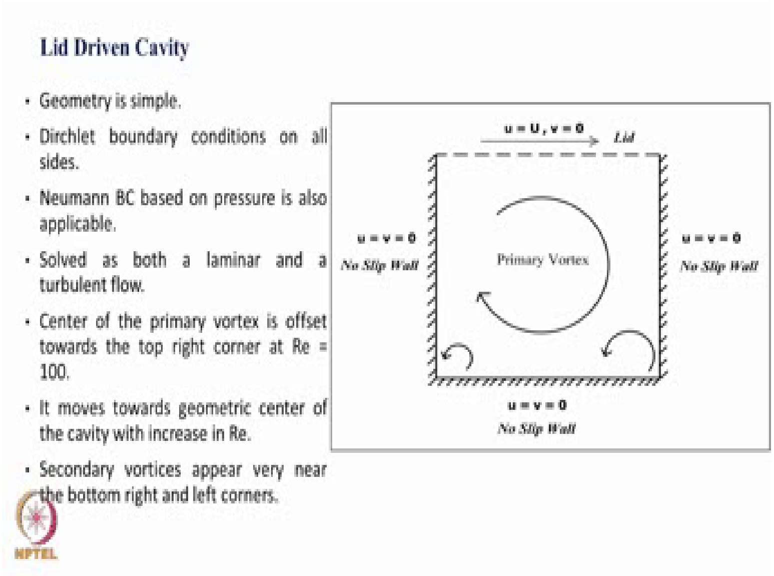
Greetings and it is my pleasure to welcome you again to this course on CFD. We are now onto the week eight of this course, and this is the last week for this course. So far, we have lean different techniques in CFD. And in this week, we are going to particular see how some of these techniques are applied for a problem. So, in this week, we will try to explain different discretization, convection and diffusion, pressure velocity coupling method, time integration procedure for a test problem. And we will also display corresponding code and how these are actually implemented in a working code. For this purpose to take a test problem what is known as lid driven cavity.

(Refer Slide Time: 01:30)



So, we will first define the problem then explain different numerical strategy to approach CFD to get a solution for this problem. We will do detailed procedure and explanation of every step, and display corresponding code. Once you get solution then we are interested to see the result in different form, and that stage of CFD is what is known as post processing. In addition to primary variable in its forms, you also have derivatives of primary variables and we will explain how to get different post processing using this primary variables and show that result.

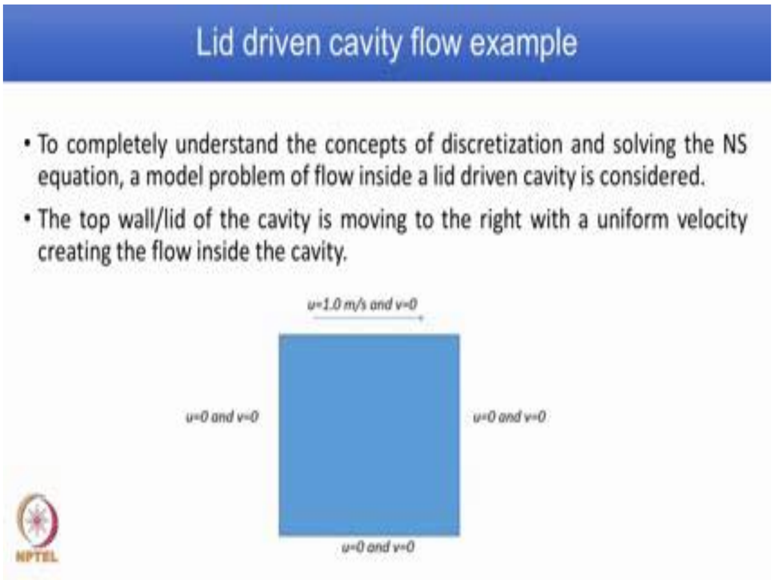
(Refer Slide Time: 02:35)



The problem considered is the lid driven cavity on here what is shown here is a 2D cavity, it can be three-dimensional also. So, in this, we have depth and then width depending on the ratio of depth to width, it can become the square cavity or cavity of different aspect ratio. Now all the three sides of wall that is have it is representation as hash line; on the top, you have the lid and it is driven by a velocity and for this particular case u is equal to u, a specific velocity is given. Now this can be one particular value or it can be a function of sin or cos. If it is two d then it becomes square cavity, if it is three d it become three-dimensional cubic cavity.

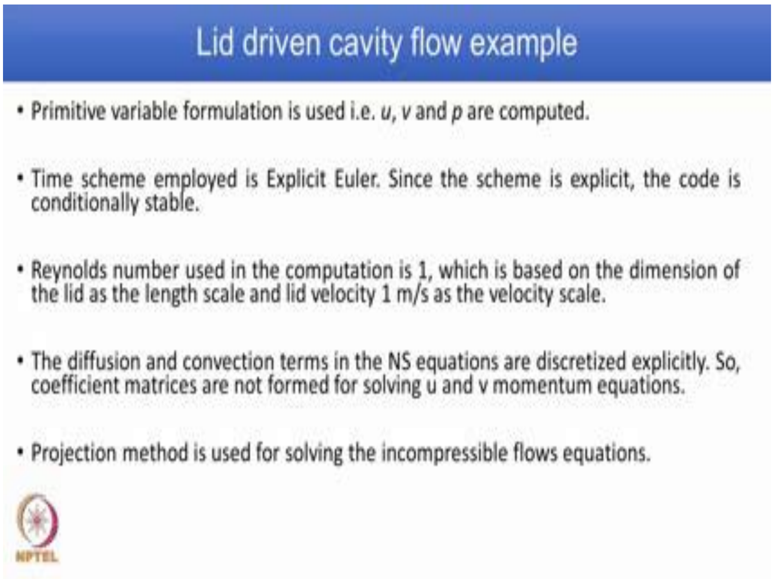
Now in this problem, the geometry is very simple and you can also understand from the figure. Once you have this driving velocity imposed on one side of the problem then you have a primary vertex form at the centre, and you have a corner vertex form at these two corners. You can have a Dirichlet type of boundary condition that is specified a particular value for the variable this is what is known as Dirichlet condition this we explain in the week one or week two lecture. So, in this figure Dirichlet boundary condition is in terms are velocity as you can see h ere u equal to v equal to zero on all the three sides of the wall. Then on the top, you have velocity driving condition u equal to u, v is equal to zero. It is also possible to prescribe Neumann type of boundary condition for pressure. Depending on the Reynolds number it may be a laminar flow or turbulent flow. As I mentioned in the beginning, you have primary vertex form and you have vertex from and these two corners. Now depending on Reynolds number a primary vertex either stays at the centre or it moves to one corner. Secondary vortices appear very near the bottom right and left corners.

(Refer Slide Time: 05:31)



The intension of this week class is to completely understand the concepts of discretization solving the Navier-Stokes equation using a model problem of flow inside a lid driven cavity. As I explained previous slide for this problem the top wall in other words the lid of the cavity is moving to the right with a specific uniform velocity and thus create the flow inside the cavity. As shown here, so u equal to 1 meter per second, it is the specific velocity given and the lid is moving from left to right.

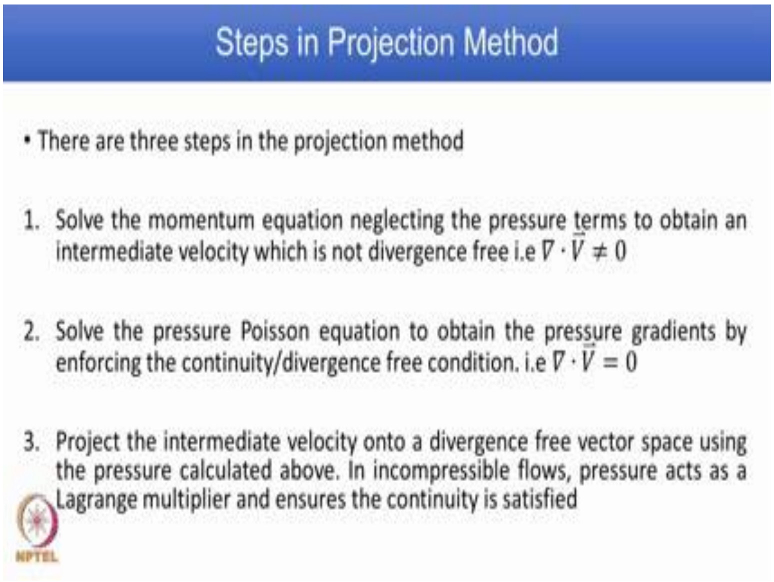
(Refer Slide Time: 06:21)



We mention you can have a vorticity steam function formulation of the governing equation or it is also possible to solve using primary variables. When you say primary variables they are u, v, w and pressure, because we consider two-dimensional situation, we have only u and v pressure is always there. For the sake of simplicity, we consider explicit Euler time integration, because it is explicit, we learn scheme is conditionally stable. Reynolds number need define, and we need to have a length scale and velocity scale. The length scale in this problem it can be a side of the cavity or it can be aspect ratio of the cavity. In this particular problem, because we are considered square cavity all the sides are equal. So, the side of the cavity can be length scale and the top lid with the driven velocity is considered as the velocity scale.

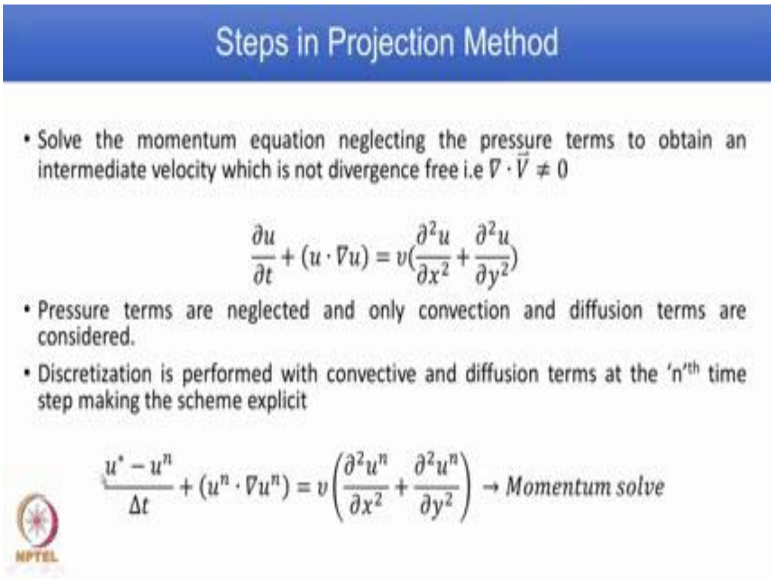
So, the Reynolds number computed based on this length scale and velocity scale is one. This is only for demonstration purpose; you can increase the Reynolds number and investigate the flow inside this problem. We have a diffusion term on the right side and convection term on the left side. So, diffusion term and convection term in Navier-Stokes equations are discretized explicitly. So, coefficient matrices are not formed for solving u as well as v momentum equations. Then we have procedure called pressure velocity coupling; we learned three four methods that is MAC algorithm, SIMPLE, SIMPLE R, SIMPLE C and projection method. In this demonstration problem, we use what is known as projection method to solve incompressible flow equation.

(Refer Slide Time: 08:46)



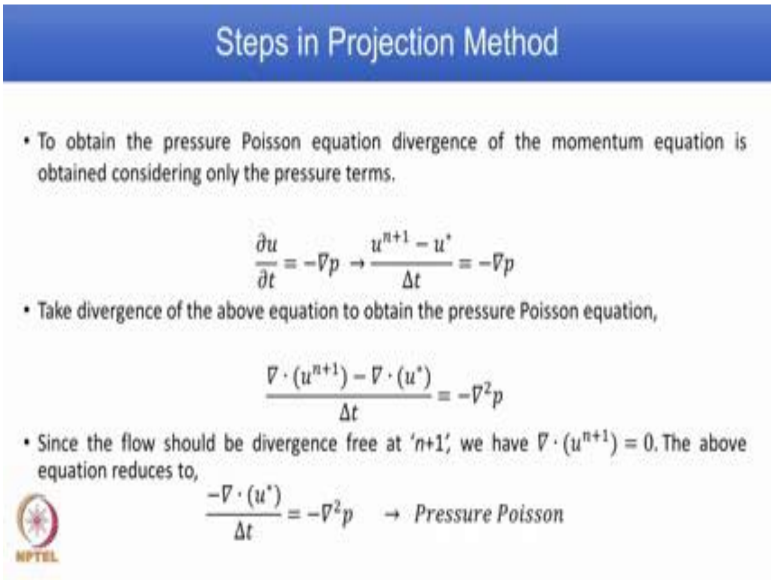
You also learnt what is projection method. However, for the sake of completeness, we repeat the steps involved in projection method. There are basically three steps; in the first step, we solve momentum equation without considering pressure term. So, in this problem, we are considering 2D situation, so you have u-momentum equation and v-momentum equation. In u-momentum equation, we have minus dou p by dou x; in the v-momentum equation we have minus dou p by dou y. So, in the projection method, we do not consider these pressure gradient term and solve without considering, hence the obtained solution does not satisfy the divergence condition that is delta dot V equal to zero. Based on this velocity we set up the Poisson equation for pressure, which is linked with continuity equation also, hence we get the Poisson equation. Once you solve Poisson equation then you get the pressure. Project the intermediate velocity onto the divergence free vector space using the pressure calculated above; and this pressure act as a Lagrange multiplier and ensures continuity is satisfied, this is important step in the projection method.

(Refer Slide Time: 10:24)



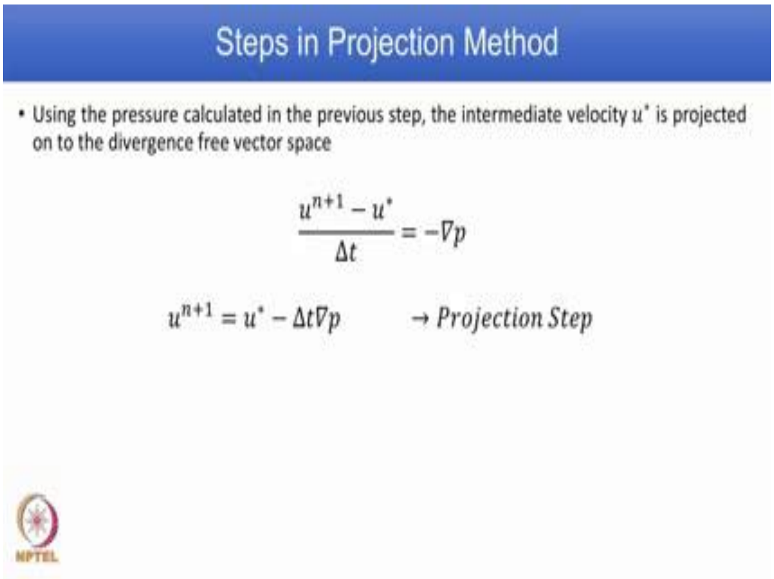
Once you solve the momentum equation neglecting pressure terms, we get equation as shown here. So, you have a time derivative term as the first term on the left hand side and then convention term as a second term on the left hand side then we have only diffusion turn on the right hand side. We are not considering any other source term hence the equation appear as simple as shown here. Pressure terms are neglected, only convection and diffusion terms are consider. Discretization is performed for convection term as well as diffusion term at nth time step, making the scheme explicit. So, the discretized equation for u momentum equation is shown here, all are evaluated at nth time level; new time level quantity to be determined is given by the superscript star, because pressure is not considered, obtain velocity is the temporary velocity field since we use superscript star.

(Refer Slide Time: 11:40)



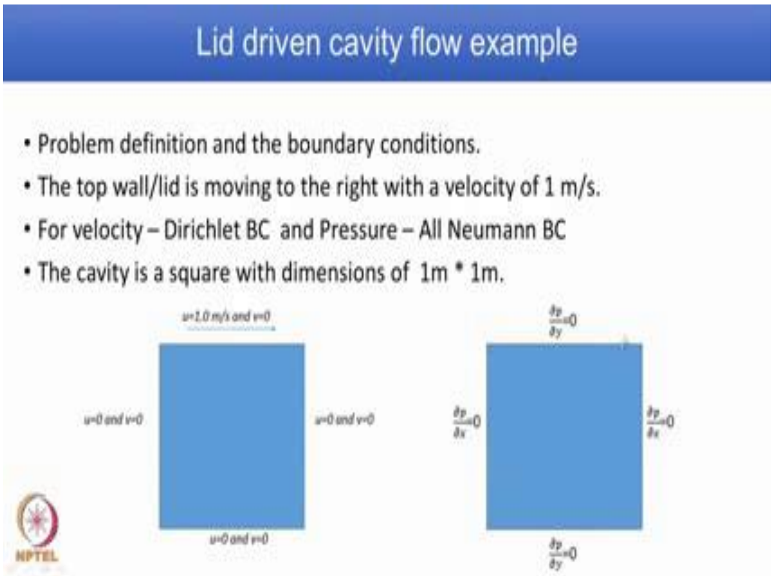
Now to obtain the pressure Poisson equation, divergence of moment equation is consider, considering only the pressure terms. So, dou u by dou t equal to minus del p and that implies u at n plus 1 level minus u star by delta t equal to minus delta p. Take the divergence of the above equation to obtain equation what is known as pressure Poisson equation. So, we do that step here as shown. So, we get minus del square p on one side, and source term for the pressure Poisson equation is on the left side. Now we can repeat all the steps for the second momentum equation that is v momentum equation. Since the flow should be divergence free, but that is what the meaning of del dot v equal to zero; in other words continuity is satisfied. At new time level n plus 1, we have the condition dell dot u n plus 1 equal to zero. Now using this condition, the above equation gets reduced to as shown here minus del dot u star by delta t equal to del square p, and this equation is what is known as a pressure Poisson equation.

(Refer Slide Time: 13:14)



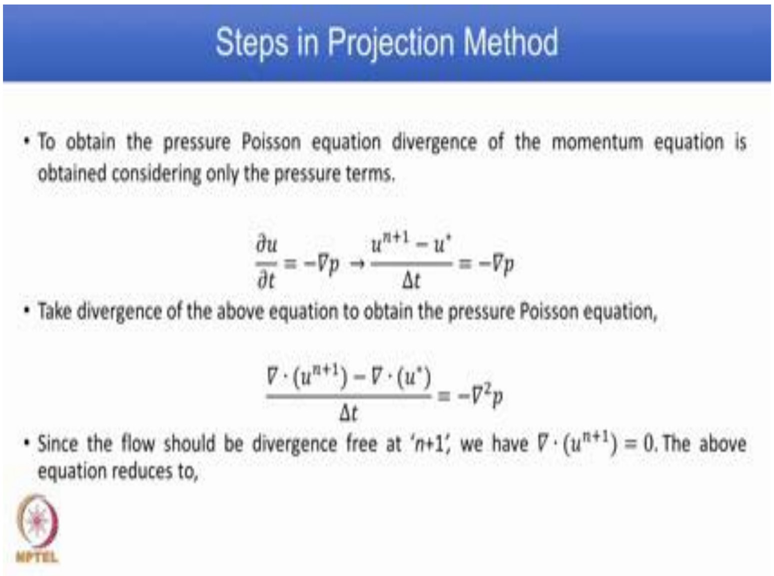
Using the pressure calculator the previous step that is one solve the pressure Poisson equations you get pressure p it is also what getting remembered that passion equation or the Laplace equation or elliptic equation. And we need to have boundary conditions prescribe on all the sides using the pressure calculator in the previous steps can project intermediate velocity u start on the divergence free vector space as shown here. So, u at n plus 1 level minus u star delta t equal to minus del p and u n plus 1 is the final corrected velocity equal to u star minus delta p and del p and this is important steps in the projection method and this is called projection step.

(Refer Slide Time: 14:17)



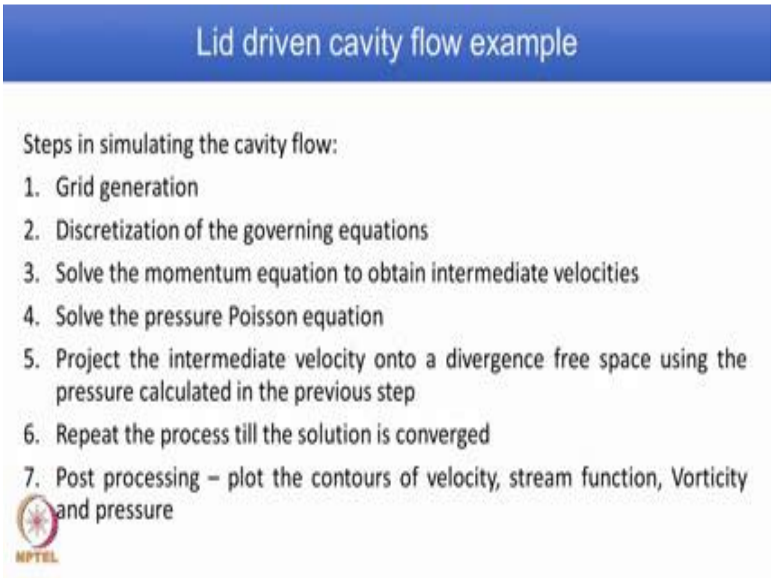
Now, we have a test case problem lid driven cavity define the problem with the boundary conditions. We already mentioned top wall lid is moving to the right the velocity of 1 metre per second. If you are using velocity to prescribe boundary condition then you have Dirichlet boundary condition or if you using a pressure then you have Neumann of the boundary condition, the cavity is square into dimensions 1 metre by 1 metre and for both velocity as well as pressure type of boundary condition that is a description given here. So, the first one is for velocity as you can observe here, the Dirichlet boundary condition applied for velocity u equal to zero v equal to zero and so on. Now if you using pressure to apply Neumann type of boundary condition and that is what shown here. So, dou p by dou x zero on this vertical spaces and dou p by dou y is equal to zero on this horizontal spaces.

(Refer Slide Time: 15:26)



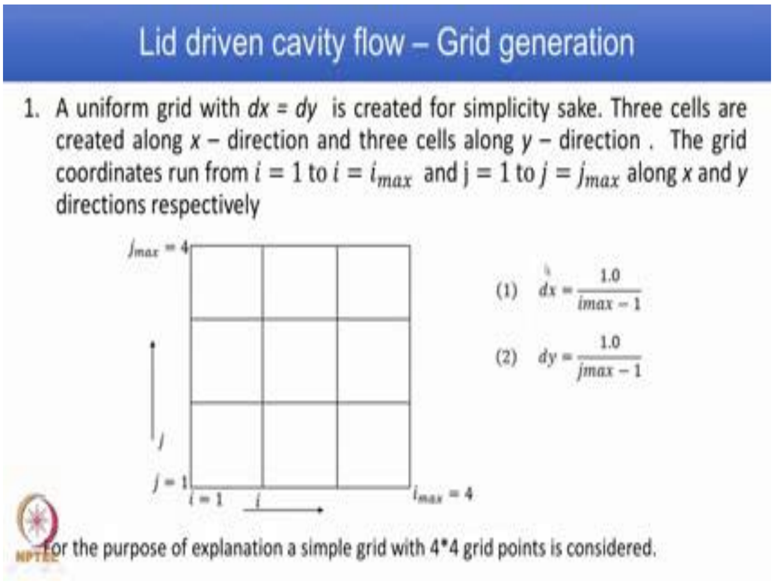
We also learn about storage of variables; we have three possibilities, one is collocated, staggered and then semi-staggered. We learnt collocated method of storing the variable result what is known as checker board problem and there is a pressure oscillation. To avoid that we have another method what is known as staggered way of storing the variables. In staggered scheme, we have the u-velocity unknown are located on the vertical faces and v velocity nodes are located on horizontal faces; and pressure is stored at the centre of the cell. The gird runs from i is equal to 1 to i is equal to i max; you specify how many grids are required in x direction and how many grid you design in the y direction. This is the simple problems, hence we use structured grid; we can use again uniform grid or non-uniform gird. In this problem, we define with the uniform grid. Staggering the u, v and pressure unknowns removes the pressure of oscillation which is a case in the case of collocated grid arrangement.

(Refer Slide Time: 16:48)



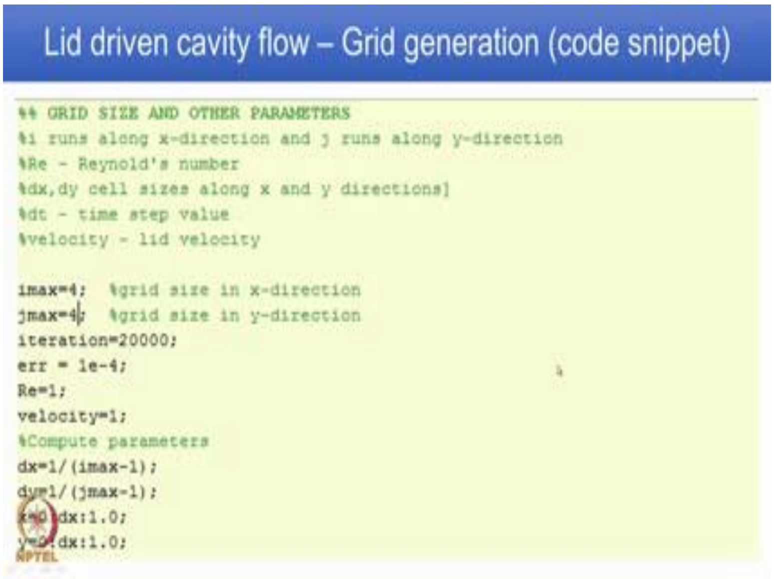
Steps in simulating cavity flow; first we do grid generation then discretize the governing equation and solve the momentum equation to obtain intermediate velocity u star, v star; solve pressure Poisson equation, project the intermediate velocity onto the divergence free space using the calculated in the previous step, then repeat the process till the solution is converged. So, we have a convergence criteria defined; based on that convergence criteria decide whether iteration needs to be stopped or to be continued. Once you get solution then you are interested in post processing. Post processing of the results can be in many forms, you can have contours, you can have a line or can go for advanced post processing, stream function, vorticity contour and so on.

(Refer Slide Time: 17:56)



So for the purpose of explanation, we take a simplified grid, structured uniform grid with the 4 by 4 grid points that is four in the x-direction, four grid points the y-direction. We define uniform grid in such a way delta x equal to delta y. Three cells are created along x-direction and three cells are created along y-direction. The grid coordinates run from i equal to 1 to i equal to i max, j equal to 1 to j equal to j max along x and y direction respectively. And this is a gird arrangement shown here. So, i is x direction, j in y direction, we define four girds in i direction, so 1, 2, 3, 4; similarly, four grids in y direction 1, 2, 3, 4; so 4 by 4 results in three cells in the respective directions. So, we have totally 9 cells. We are going to follow finite difference method of solving or discretization the equation, and we also mention staggered grid is used to store variables. So, dx and dy are uniform, because we define number of grids in x direction and y direction to the same, that is 4 by 4 in this case, we can find out what is d x and d y.

(Refer Slide Time: 19:40)



So, this is the corresponding code that is used to generate grid. So, we have a command; gird size and other parameters; i run along x direction, j runs along y direction. R e is Reynolds number; dx dy cell sizes along x and y direction; dt is a time step value velocity is actually lid velocity. We already defined i max to be four actually change into any number; this is a number of grid size in x direction similarly for y direction j max equal to four this is the grid size in y direction. And this is required in later, I am now showing you actual working code written in Matlab. So, what is shown here is a beginning of that code. So, some parameters needs to be defined beginning of the code and this one such parameter what is shown here is the iteration that is given as numbers twenty thousand and this is the tolerance limit or convergence limit that is error one ten to the power minus four; Reynolds number is 1, velocity is 1. So, in this particular slide, we are interested to see how the grid is generated and what is a corresponding code available to generate a grid.

So, we have compute parameters, then dx equal to one by i max minus one and d y equal to 1 by j max minus 1. We decide number of grid lines in the particular direction that is given by i max and j max, then the starting point for x for grid in the x direction. So, x equal to 0, it goes to 1, and with the delta x it that is defined as dx. Similarly in y direction, it starts from y is equal to 0, and n at 1, because we have define the side of the square cavity as 1, and we have also explain the dx equal to dy; for simplicity we are considered dx is equal to dy. And in this particular line, we have y is equal to zero, dx it can also be dy and starting from y equal to 0 to y equal to 1.0. In this module one of this week, we have explained the test case problem and then we have decided on discretization procedure, pressure velocity coupling, storing of variables, grid arrangement then I displayed actual working code, how the grid is generated. In the next module, we will go to next part of the algorithm where we talk about discretization of convection terms and diffusion term.

Thank you.