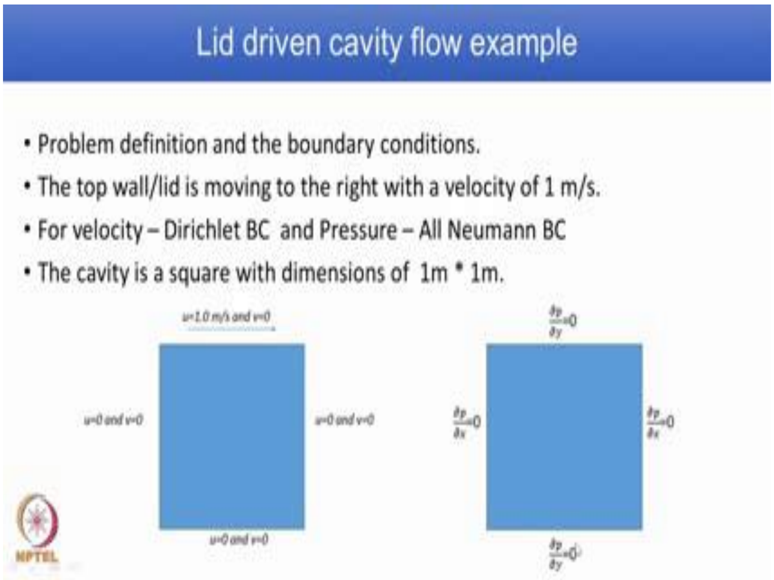
Foundation of Computational Fluid Dynamics
Dr. S. Vengadesan
Department of Applied Mechanics
Indian Institute of Technology, Madras

Lecture - 44
Assignment

Greetings this particular video is about assignment that we are giving for week seven and week eight. So far, we have given objective type assignment, whereas in week seven we have given programming assignment. Week seven forms matrix solution procedure and it is necessary that you practice them. Hence, we have given a programming assignment, there are three languages possible C, C plus plus and Python. We have given four problems one based on gauss elimination method, second problem based on TDMA procedure, and third problem based on LU decomposition and fourth problem based on iterative procedure. Coding language is part of this course, hence you must know any one of these languages and start programming and become part of the CFD community.

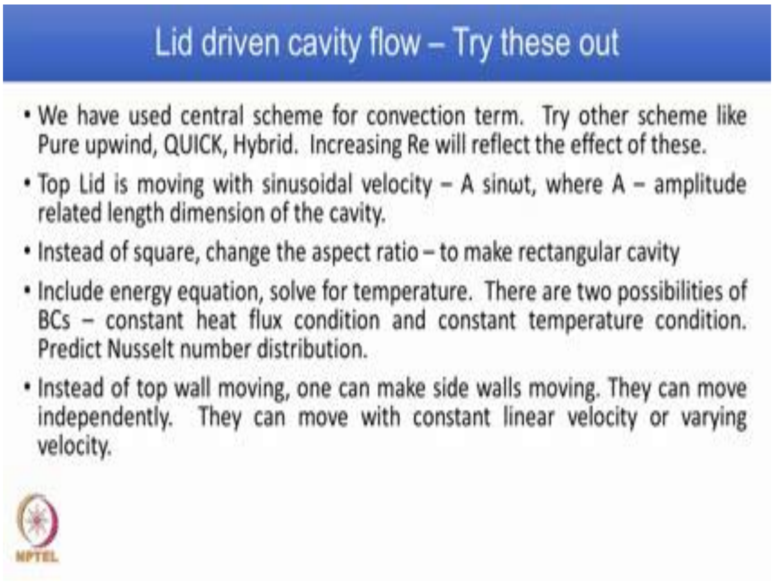
In the week eight, we discussed in detail a test case problem that is flow in a lid driven cavity, and we have demonstrated with the help of a mat lab code. We are also providing you the complete working code as a separately; and to strengthen further this knowledge, we are giving you advice to practice more problem. So, this particular video is talk about what all the problem that you can try with this knowledge, you are most welcome to use this mat lab code as it is or you convert this mat lab code to any language that you are comfortable with; for example, C or C plus plus and Python. In our regular CFD course, this is how we practise. All the students are encouraged or given assignment to start coding write from the beginning itself. For example, we will start writing the diffusion equation then there is slowly build the code as we proceed with the syllabus content. We are trying to do a similar one in this exercise also. Now I am going to explain something more about assignment that we wanted to practice for week eight.

(Refer Slide Time: 02:48)



So, the problem that we are considered the week eight was lid driven cavity example. We define the problem, we define the boundary condition, we mention the top wall or the lid is moving from left to right with the specific velocity condition. And for velocity, we used Dirichlet boundary condition; and for pressure, we used Neumann boundary condition. The cavity in two-dimensional situation is a square' if it is the three dimensional situation it becomes a cube, with the dimension of 1 metre by 1 metre by 1 metre. And this is the illustration we used to explain problem as well as the boundary condition implementation. So, on the left, you see explanation for velocity boundary condition; top is the lid moving with the velocity 1 metre per second, and v equal to 0, and all the three sides velocity equal to 0, of a pressure on all four sides we have Neumann type boundary condition. So, accordingly in the x direction it is dou p by dou x equal to 0; in the y direction, it is dou p by dou y equal to 0.

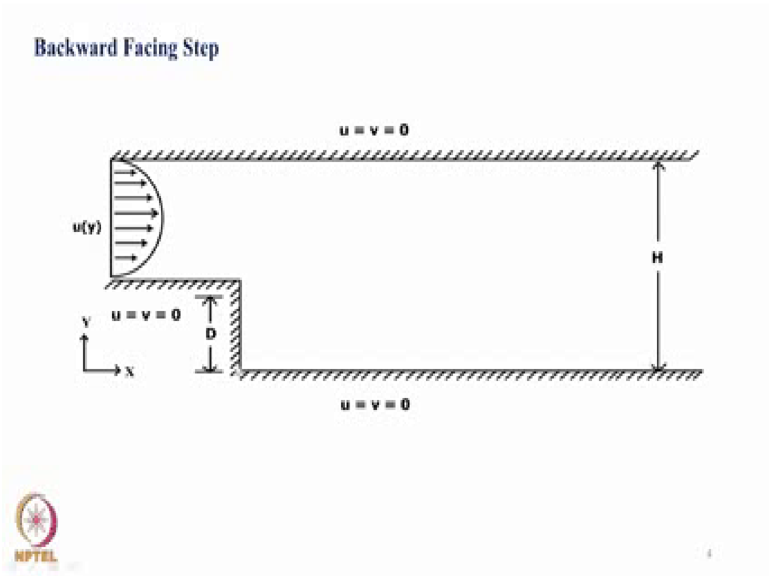
(Refer Slide Time: 04:11)



Now, based on this, the same problem can be extended and you can try anyone of this problem. We have used central scheme for convection term. If you can recall in week five, we taught different other schemes for example, pure upwind, QUICK scheme, hybrid scheme. Increasing the Reynolds number will have the reflection on effect of these schemes. Hence, you can try changing the code from central difference scheme to any one of this schemes, and try increasing the Reynolds number to understand yourself the behaviour of these schemes for different Reynolds number for the same set of mesh. In the problem demonstration, we have used top lid moving with a constant velocity for example, u equal to 1 meter metres per second. It is also possible to oscillate the lid with the sinusoidal velocity that is a sin omega t where a is an amplitude, which is related to some major length scale related to the problem for example, length dimension of the cavity and omega is the frequency you can try with different compound. So, this will give you not linearly moving lid, but it will give oscillating lid.

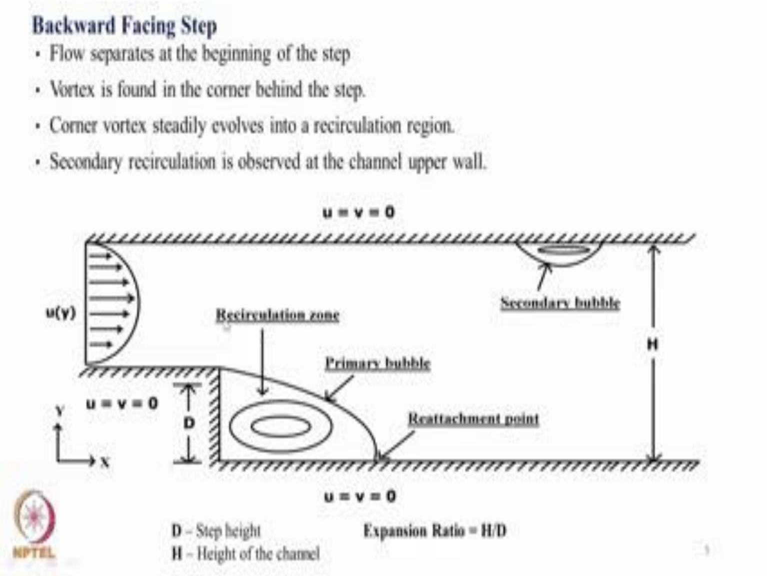
So, instead of square you can also try cubic cavity, you can also try changing the aspect ratio, so it becomes deep cavity or shallow cavity depending on the definition of aspect ratio and depending on value of aspect ratio. So, aspect ratio means depth to the width. And we explain the problem with only primitive variable that is u momentum equation, v momentum equation and continuity equation. You can also include any other scalar equation, for example, you can include energy equation and solve for temperature distribution. Here again there are two further the possibilities for boundary condition, one you can use constant heat flux boundary condition, other one constant temperature boundary condition. Then predict Nusselt number variation and understand the thermal characteristics associated with this flow. Another possible variation, we explained the problem with top lid moving, it is not necessary only top lit move, you can also have another side wall also moving; it is possible that they side wall move with same direction of the velocity or they can move into opposite direction of the velocity.

(Refer Slide Time: 07:00)



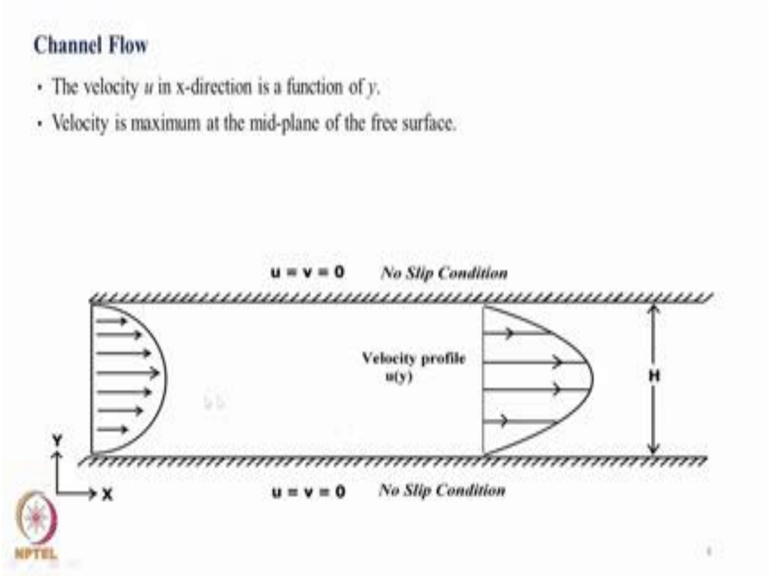
Next problem that we advice to try flow through a backward facing step we explain this problem much earlier. However, I am interested to explain that problem again. So, the inflow is on left side, you can have a fully developed flow velocity profile explained as given here u of y with some expression as a parabolic expression. Then on the top is a wall with a boundary condition u is equal to 0 and v equal to 0, then you can define different steps height D and get to know the effect of step height on the flow parameter then you have this as the wall, the side step is also a wall and then bottom is also a wall.

(Refer Slide Time: 07:46)



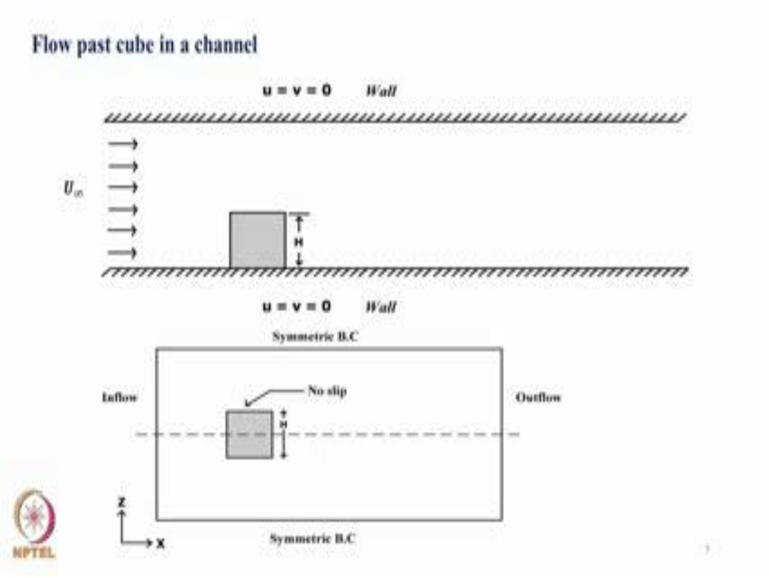
We understood the behaviour associated with this problem, flow separates at the beginning of the step as I am showing here, then it comes back at reattaches on the bottom wall as I am showing here. Then there is a eddy or re circulating zone, prediction of the recirculation zone is very important and especially in the case of you deal with temperature or energy equation then prediction of skin friction along the wall is also important. We also know there is a corner vortex that is formed and slowly it becomes a re-circulating region depending on the Reynolds number. There is also a secondary circulation form on the top wall as I am showing here. This again depending on the Reynolds number, hence you can try increasing the Reynolds number, and observe yourself whether you are getting a secondary bubble on the top.

(Refer Slide Time: 08:49)



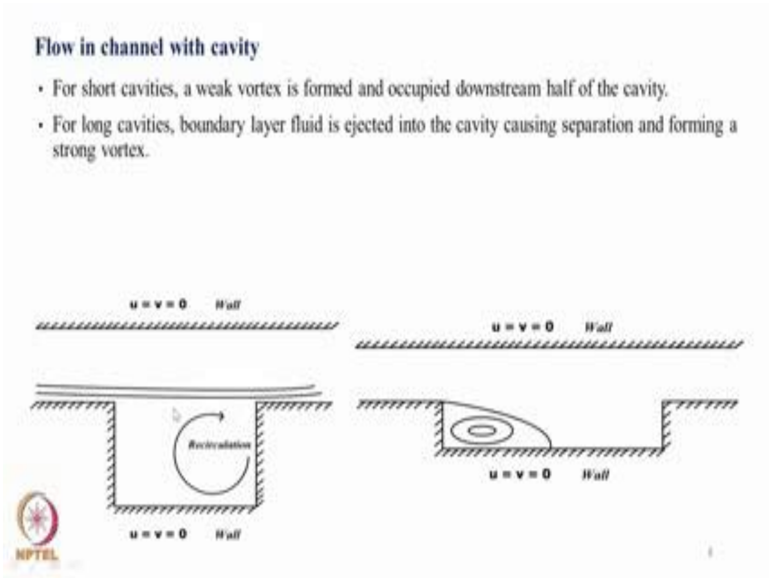
The third problem that we can think of is flow in a contour. So, flow through a circular cross section pipe as I am showing here. You can give at the inlet either the parabolic velocity profile as it is described here or you can describe a uniform velocity at the inlet. If you are giving a uniform velocity at the inlet, it will take a long channel for the floored develop and become a fully developed velocity profile as it is shown in here. If you are giving already a parabolic profile, which is very close to the fully developed condition then you need only a smaller length and flow will develop. Again boundary conditions are given u equal to v equal to 0 on both the side wall. So, velocity u in a x-direction is a function of y and velocity is maximum at the mid-plane of the free surface.

(Refer Slide Time: 09:46)



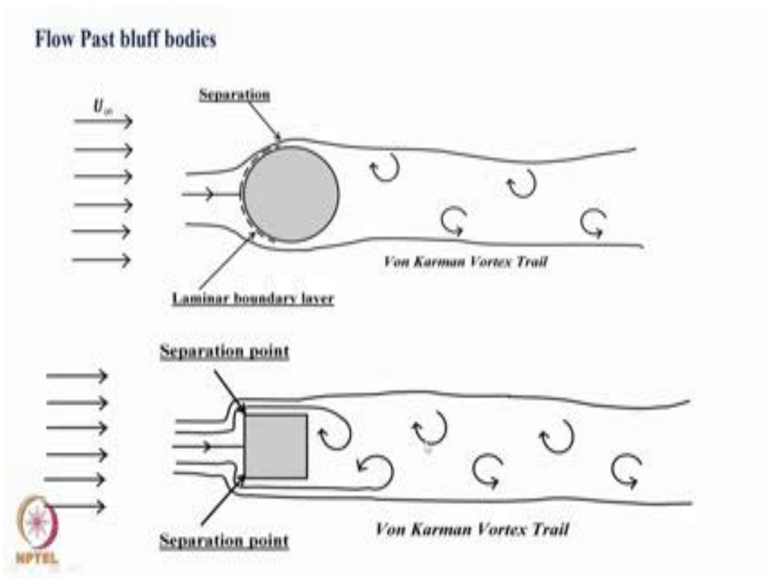
Next problem you can think of is flow past cube in a channel; it is a three dimensional problem. So, one view is shown here, the top view is shown in here; the cube is actually placed in one view as I am showing here, and from the top, you get to see figure as shown here. Once again you can have a variation in the inlet velocity condition, either you can prescribe the uniform velocity as I am showing here or you can also prescribe a fully developed velocity profile as we did in the previous example. Prediction of this flow is very interesting, we also observe the flow understand presence of the body and separates at this point and flow go around the geometry and create different interesting fluid dynamics phenomena behind the geometric.

(Refer Slide Time: 10:42)



The next one flow in a channel with the cavity the first problem that we discussed is a closed cavity, it is also possible to have a cavity has part of the channel and that is what you are seeing here. So, the top is a wall, and this is a channel, suddenly you are experiencing a cavity; it is like combination of cavity and backward facing step. So, for short cavities, a weak vortex is formed and occupied downstream half of the cavity; and for long cavities, boundary layer fluid is ejected into the cavity causing separation and forming a strong vortex. So, you have two such possibilities.

(Refer Slide Time: 11:28)



The last problem that we are advising you to try is external flow. So far, we have done internal flow, it is also interesting as well as important to know how schemes behaves differently for external flow. External flows we are discussing two important problem flow for circular cylinder, flow for square cylinder, the two problems are given. So, if you have the code written, then it is only changing the geometry from circular cylinder to square cylinder. The difference is in the case of circular cylinder separation point shown as shown in here it is not fixed it changes according to the Reynolds number and that dynamic change of the separation point results in different wake pattern. Whereas in the case of square cylinder separation points are fixed whatever the Reynolds number and that gives a different wake pattern.

So, as far as week eight is concern, I repeat again we are sharing you mat lab code. You are most welcome use that code as it is or you can take that code format and write your own code in whatever language you are comfortable with. And try all these problem, there is no submission it is only to increase your confidence level and for better practice as well as to understand behaviour of different schemes. We will have separately on small assignment as in the previous weeks a multiple choice question, but that is not that much learning unless you practice CFD with your own code.

Thank you.