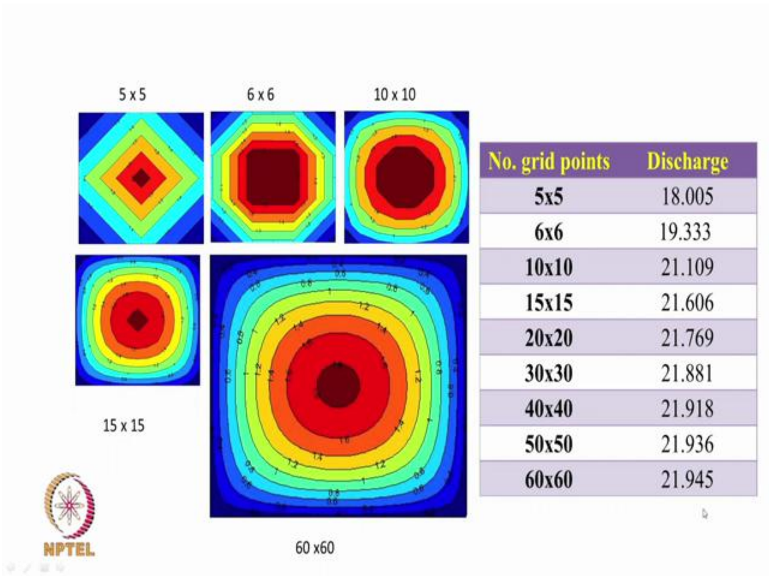
Computational Fluid Dynamics
Prof. Sreenivas Jayanti
Department of Computer Science and Engineering
Indian Institute of Technology, Madras

Lecture – 10
Effect of grid spacing & upcoming course outline

We have done manually both the problems that is flow through a rectangular duct and also flow through a triangular duct. Because we are doing manually we had to restrict ourselves to small number of cells, but you could write a computer program and then increase a number of cells so that you would actually get a proper solution and that is what is done by a student in a regular CFD course here at IIT Madras and the results are shown here.

(Refer Slide Time: 00:38)



No. grid points	Discharge
5x5	18.005
6x6	19.333
10x10	21.109
15x15	21.606
20x20	21.769
30x30	21.881
40x40	21.918
50x50	21.936
60x60	21.945

What we are seeing here are the velocity contours for the rectangular flow case with 5 by 5 cells; 5 in the x direction, 5 in the y direction; 6 in the x direction, 6 in the y direction; 10 by 10 here; 15 by 15 and 60 cells in the x directions, 60 cells in the y direction; all done using the same program so you need to be systematic in understanding how to do it and how to code it, but it is not difficult to write the computer program and if you can write that then you can easily do these kind of things, you can run it for any number of divisions and then you can get a solution and what we see here is that as we increase the number of cells here, that is number of grid points then the contour lines become very smooth.

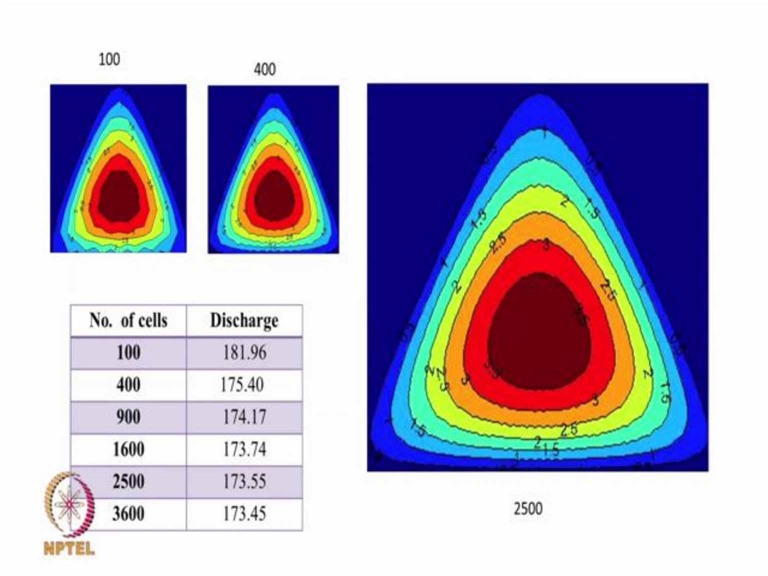
In all cases you see that the maximum velocity is at the center, the velocity value here is given in terms of color coding with their different color contour values are given here and the counters obtained with 60 by 60 divisions here is shown and you can see that it is very reasonably looking velocity profile is what velocity contours have what we see. In the center we have the highest velocity and then as we go out towards edges then we have lower and lower velocity and this is what we precisely expect, but in order to get this kind of smooth variation we had to have large number of cells here.

So, this is one of the features of CFD that is you need to have a large number of cells in order to get a good solution and so it is not fit for hand calculations only if we can do computer based calculations you can do CFD. Another we are looking at the results is shown in this table here and here we are looking at the total discharge that is volumetric flow rate through this duct cross section, that is you get the cell values at this 25 points here and then you just add them up or you can do weighted average based on the area of each tile and then you can get a total flow rate here. The numbers do not matter it depends on what size they have taken and what pressure gradient they have taken.

But in all these cases, the pressure gradient; the constant c is kept the same value and dimensions of the duct are also same, the fluid is also the same. What is being changed is the number of points here and we see that as you increase the number of points this value which is 18, 19.3, increasing, increasing, but as you increase further it is becoming constant. So, here there is 21.9, 21.93, 21.94, so it is; in a way it is becoming constant value. So, there would not be significant difference between success values as you increase the number of grid points and that is when you can claim that you have a grid independent solution and we expect the gradient kind of solution to match with the correct solution; exact solution equation.

In cases like this, where there are modeling arrays are very small and the approximations that we are made in doing the discretized equations, we said that we will give values at only grid points. When we compute the flow rate through this, we multiply the grid point velocity; times the grid area, the cell area; so there is an approximation there. So, it is not a analytical integration, it is approximation that is introduced, but as you increase the number of tiles, as you increase a number of grid points; then that error becomes less and less and we see that happening here.

(Refer Slide Time: 05:01)



No. of cells	Discharge
100	181.96
400	175.40
900	174.17
1600	173.74
2500	173.55
3600	173.45

The same thing can also be done for the triangular duct and here again we have with 100 cells, 400 cells, 2500 cells and what we did in hand calculation was just 9 cells. Again you have to do coding and you have to be good at it in order to get it going, but it is possible, it is been done by students like you and if you do that you can see again as you increase the number of grid points, you get smooth contour lines, again you have at the center we have the highest velocity and as you go towards the corner, the velocity decreases and nice and smooth kind of things are we are getting here, there is some small variations here those are the kind of errors that may be expected.

If we increase a number of grid points, if we increase the accuracy of the calculation even those may disappear and again the total discharge that is calculated through this cross section; for the same c value and for the same cell dimensions in all cases, the same duct dimensions that just by increasing a number of cells, we can see that is gone from

182 to 175, 174, 173, 173.7, 173.55, 173.45; it is converging to a value and that again is a characteristic feature of CFD. So, with these two examples and using two different approaches which is the finite difference approximation and the finite volume approximations, we have seen how we can take a partial differential equation and then convert into algebraic equations and then solve them using a certain methods; any methods, we have used as a certain method. We can get solution which results in contours like this velocity contours like this.

So, this is essentially what CFD is, but this is not only aspect of CFD because what we have done is take the simplest possible case. In actual case, we have lot more difficulties that we encounter, the equations that we have taken are only Laplace type or Poison type of equation; so in that sense it is the probably the simplest form of the governing equations that we can find in fluid flow. The actual governing equations are much more complicated and it is not; we have solved only one partial differential equation in actual case you will have may be 2, 3, 4, 5, 8, 10 whatever number of equations that may be there, even hundreds if you do chemical reactions and then if you have a large number of species then you have hundreds of reactions are also possible, hundreds of equations are possible

So when you are dealing with those many number of equations and when you are dealing with the nature of the equation which is different from the simple case that we have taken here, it is a boundary value problem and it is a relatively easy problem, so for all those things you have a lot more complexity that arises in the actual CFD solution and that is what we are going to do in the rest of the modules. In the next module; this is the first module in which we have looked at essentially the skeletal structure of CFD approach and done for two different cases using two different approaches, but both are essentially following the same route of converting a partial differential equation into a set of algebraic equations and using a special method to solve those algebraic equations and getting the final solution.

So, the same kind of approach will be discussed in much more thoroughly and we bring in a whole set of new concepts which are needed to get a solution, a satisfactory solution for the real case; we can see that simplistic approach is that we have used here cannot be universally applied ok. So in the next module, we will try to look at what are the equations that would actually appear that we need to solve when we are dealing with the general case of fluid flow involving for example, flow heat transfer and chemical reactions. So, we will try to establish these equations from fundamentals so that you have a good understanding of this and we should keep in mind that the accuracy of the CFD solution is only as good as the accuracy of the equations that we need to solve and so we need to know what kind of equations we are solving and why we are solving these equations. Along with these equations, we also need to have boundary conditions and initial conditions because we have partial differential equations.

So, what type of boundary conditions is appropriate and how to specify these is also something that will see in the next module. So, by the end of the next module, we will have a good understanding of what equations we would like to solve and we already have a fairly comprehensive idea of the CFD approach in terms of discretization and all that, but we would like to put that on a more firm fitting and then we will look at what is meant by this finite (Refer Time: 10:41) approximations and what kind of approximations are possible and what kind of approximations are good. So, these are the concepts of specific to the solution of fluid flow equations, which are needed to be understood in order to get to a proper discretization scheme for a given equation.

So, in the first part of the third module; we look at the basics of finite different methods and then we look at certain concepts like a consistency stability and convergence which will dictate the goodness of the solution that may be expected and in the second part of the module we look at certain tests that can be done to verify the consistency and stability and all that and we will finally, come up with a good way of a template for the solution of a typical partial differential equation that we encounter in a fluid flow. So, that would be the end of the third module, but as we mentioned earlier; fluid flow equation is not about solving one equation, it is solving a set of equations and these are coupled equations so we need to have specialized methods for the solution of this coupled equations.

So, in the fourth module, we look at the specialized methods that have been developed; based on this fundamental understanding of the fundamental concepts of this CFD, we look at some one or two special methods that have been developed for compressible flow cases, where the equations have a certain character and what kind of other special schemes that are needed for in compressible flow, which very often occurs in practical applications, but unfortunately we would not be able to apply the compressible fluid flow type of approaches for incompressible flow prima facie in a naive way.

So, we would look at some special methods that have been developed for incompressible flows. So, at the end of this module, the fourth module; we will have known what equations to solve and what is the best way of discretizing these things and then what is the best way of coupling all these things together so that we can hope to get the solution of the entire set of equations. But we have seen today, that knowing is one thing, but having in order to get a good solution we need to have large number of grid cells and so that means, that you should be solving these equations with a huge number of grid points. Sometimes it can be ten thousand, a million is also not unhide of; when you are looking at million grid points, then your quotient metrics will be million by million. So, you need to have a very efficient method for the solution of these equations; linear equations.

If we look at conventional methods, they would not be really good, they will take so much time that you cannot hope to get a solution; in your realistic life times of your assignments or even of your industrial problems, even or of you career and all that. So, we need to have specialized methods and in the fifth module, we look at the specialized methods that have really brought a CFD solution within the reach of an engineer, who can pose the problem on the evening and hope to get a solution with the million grid points the next day morning. So, in overnight solution is made possible; only with the development of these specialized methods for the solution of the linear equations, for the solution of a w equal to b when a is very large, which has large number of grid points.

So, that will complete for us the entire spectrum of the CFD things at a basic level so we would know what equations to solve, we would know how discretize them and we would know how to couple them and we would have efficient methods for the solution of these algebraic equations so that we can get a solution like what we see for the triangular duct here in a more complicated way. But this method; what we are going to develop in the next four modules, is based on essentially the Cartesian coordinate system; for the simple case like the first case of rectangular duct. When we want to tackle more difficult geometries, irregular geometries or which cannot be put fitted into a simple Cartesian system or into cylindrical system or spherical system like the triangle that we have seen then it is important to it is usually necessary to go for the finite volume method and then we will in the last module; in the first part of the last module we look at a systematic introduction to finite volume method, what we have done is in adequate for this specific case of equilateral triangle we have done it.

We look at a more general way of doing it in the finite volume method and in the second half of the last module we will also look at how to treat turbulent flows because what we have done is for lamina flows and turbulent flows are a different based altogether; in terms of the complexity of the physical phenomena and you have a fairly comprehensive treatment, approximate treatment that is possible for turbulent flows and that is what we will be discussing in the last part of the last module. So, with that we will have more or less basic understanding of how to do CFD for essentially single component flows, single phase flows.

So, that is outline of the whole course and from next week onwards we will move into the second module.