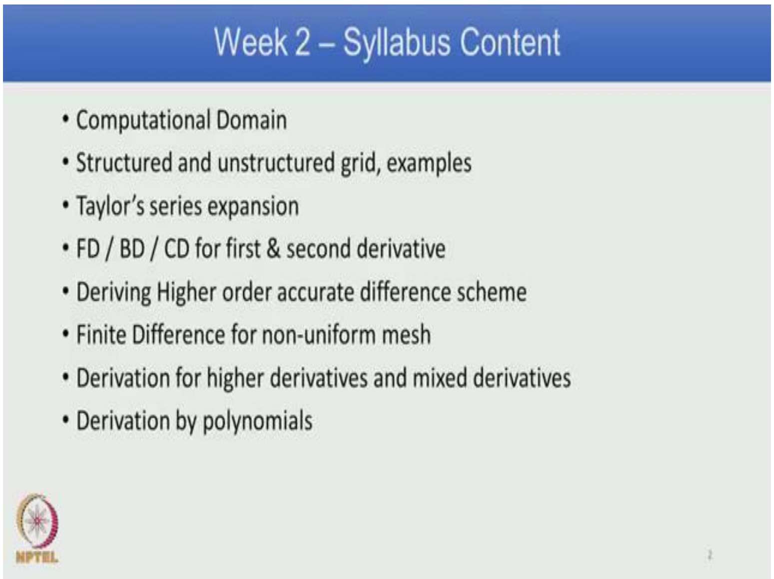
Foundation of Computational Fluid Dynamics
Dr. S. Vengadesan
Department of Applied Mechanics
Indian Institute of Technology, Madras

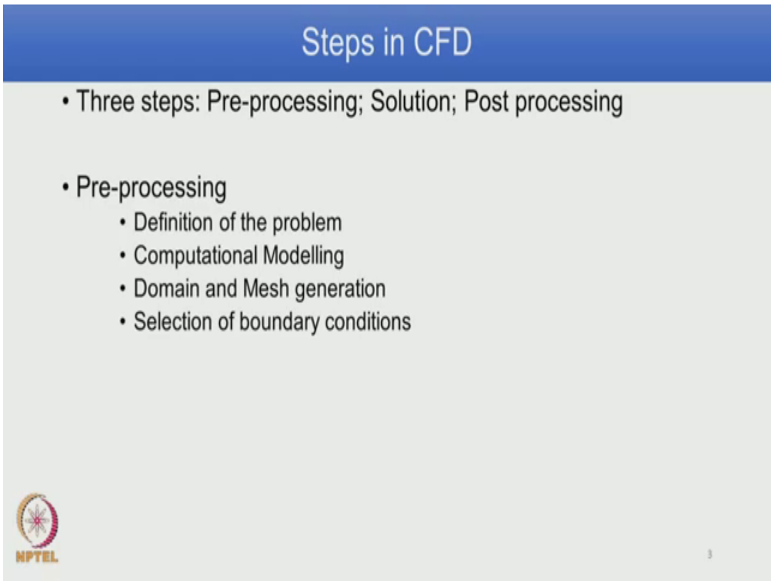
Lecture – 06

(Refer Slide Time: 00:35)



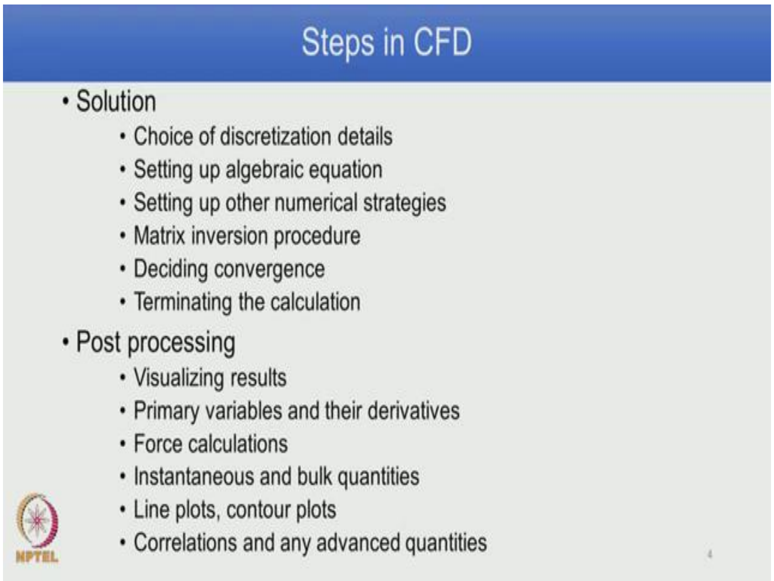
Greetings you all; last week, we had seen governing equations, boundary conditions and some test cases. We are now onto the week two, and this week, we will see important topics, computational domain, grid definitions and examples, Taylor's series expansion, how to get different forward, backward, central difference schemes, and higher order accurate scheme, finite difference for non-uniform mesh, derivation of higher derivatives and mixed derivatives. We will also see how to obtain finite difference formula by polynomials procedure.

(Refer Slide Time: 01:08)



So, in general, there are three steps in CFD; pre-processing, solution and post processing. So, in pre-processing, we just define definition of the problem, physical modeling of the problem, then computational modeling. We can simplify the physical problem and do a modeling then decide the domain, how much it should go in x-direction, how far it should take in other directions then decide the type of mesh, choice of boundary condition depending on the problem.

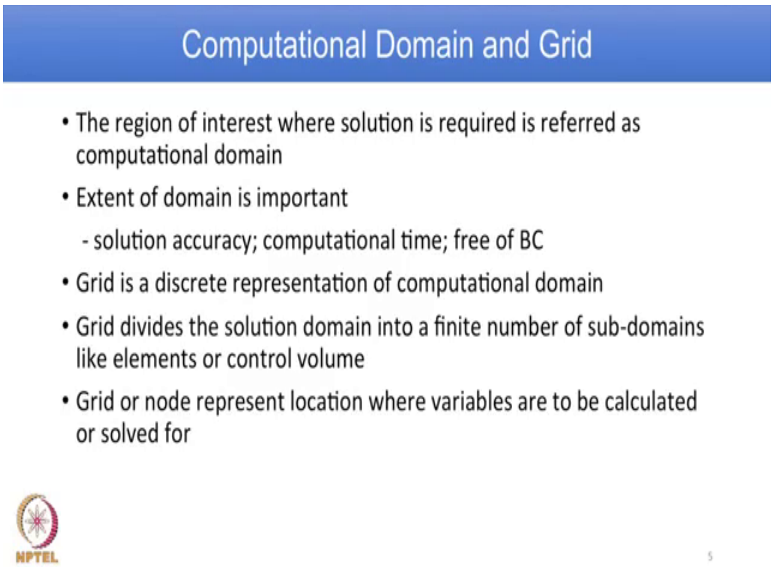
(Refer Slide Time: 02:02)



And in solution, we decide the discretization methods, then once discretize the governing equation, we get algebraic equation. We also setup few other numerical strategy, we are going to look at that later. Then we invert the matrix, so there are again different ways of doing matrix inversion. There if a solution progresses, where to stop; how to decide the solution is reached the desired condition that we are looking for that is the deciding convergence. And so decide terminating the simulation. So once we obtain solution, then what are we looking for the post processing. So, you can visualize the results, you can visualize or plot only the primary variables; so if you are solving for velocity u, v, w are primary variables. You can also plot the derivatives of stream function is a derivative, vorticity is the derivative.

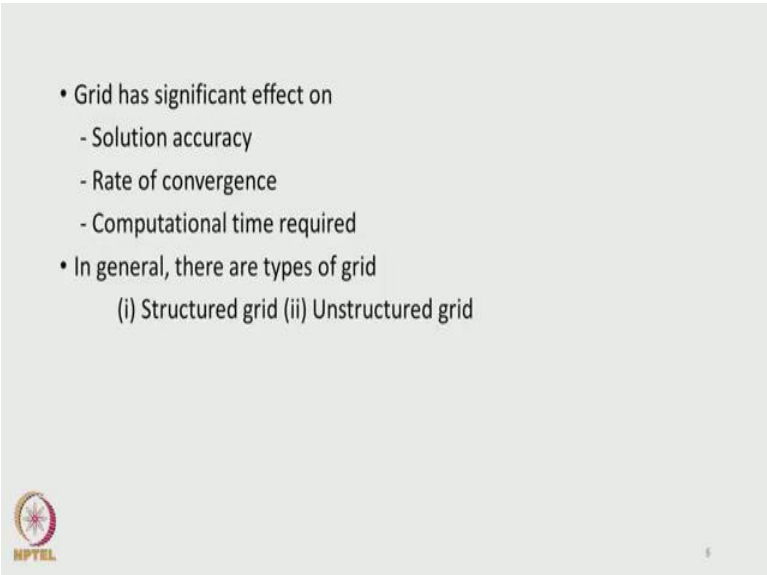
You are also interested to find out say for example, force exerted or thrust developed that also can be estimated. And if you are solving unsteady problem, then you can obtain all these instantaneously or you can average and get a bulk quantities. You can represent the results either in line plots or contour plots or super imposed one over the other plot of kind of plots. Then post processing initially we do a simple form, you can also go to the advanced stage, like find out the correlations or where the minimum, where is the maximum etcetera, so these are all stages in post processing. So, we understand there are three main stages; pre-processing, solutions and post processing. So, in this course, we are going to particularly look at sub topics in each of these three main stages.

(Refer Slide Time: 04:26)



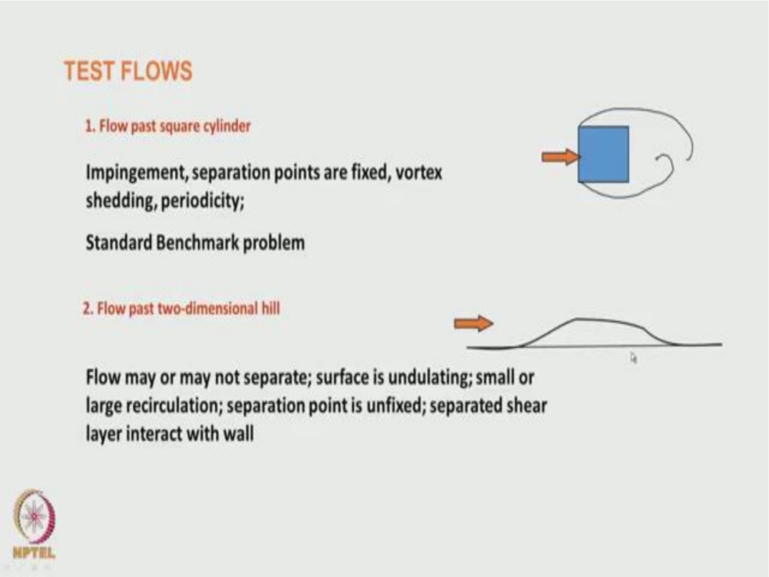
The first one is the pre-processing. We mentioned that domain where you are solving needs to be specified, and this is known as a computational domain. By definition, it is region of interest where solution is required and extent of the domain is important. So, how far you are going in particular direction is important. The domain extension affects the solution accuracy, computational time requires and the boundary condition enforced should not influence the solution. So, the problem of interest and the boundary condition implementation should not influence or affect the solution in the region of interest. Grid is a discrete representation of the domain. So once you decide the domain, then you need discretize or you need to generate mesh where you are solving governing equations. Grid divides the solution domain into number of finite number of sub-domains or elements or control volume. So, grid or node represents location where variables are to be solved for.

(Refer Slide Time: 05:59)



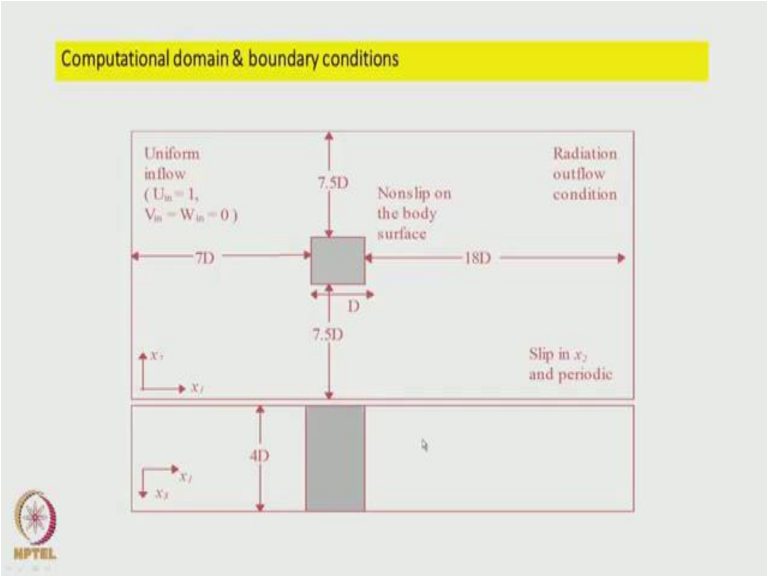
Just like computational domain grid also influences solution accuracy, rate of convergence and computational domain computational time that is required to solve and get the answer. In general, there are two types of grid, one is called structured grid, other one is called unstructured grid.

(Refer Slide Time: 06:25)



So, I am just trying to explain with help of two example problem. So, what is shown here is flow past square cylinder. The shaded portion what you see here is the square cylinder. Flow is coming from left and impinges on the surface then we already seen in square cylinder separation points are fixed and there is shear layer separation and vortex shedding happens. There is a periodic periodicity associated the shedding, and frequency of that vortex shedding is also important. This is standard benchmark problem. Another case is flow past two-dimensional hill. Again incoming flow is from left, this has slope and depending on this slope, the flow may separate; if it separates, it comes back and reattaches downstream somewhere here that depends on the slope of the front portion. If it is not separating, then it follows the geometry and then flow afterwards. This geometry also can be sort of undulating surface.

(Refer Slide Time: 07:57)

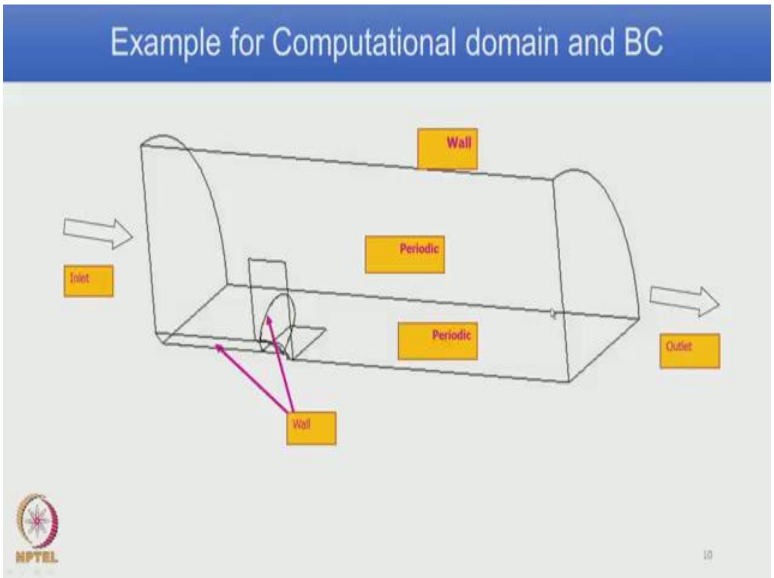


So, first problem, square cylinder is placed, and it is three-dimensional simulation. So, we have one view in x 1, x 2 plane; x 1 refers to x direction and x 2 refers to y direction. In other view, x 1, x 3 are x z plane. The flow is coming from left to the right, and we specify inlet condition uniform inflow U n equal to 1, and V in and W in equal to 0. So, the other two components of velocities are 0, only U velocity is 1. And the characteristic dimension for this problem is diameter of the cylinder, which is given here as D. So the domain, computational domain is represented based on this characteristic dimension D, so you go in this problem what is shown here is the 7D, from this side of the cylinder, inlet domain is placed at 7D; similarly the exit is on the right more side, which is placed at a distance of eighteen D from this surface of the cylinder.

Similarly, you can prescribe the location of the boundary in other two direction that is x 2 as well as x 3. So, now the question is how this 7D is decided. Why not it is 9D or 5D; similarly for other dimensions; so this decides the solution accuracy and there is particular exercise that one has to do before arriving at extend of the domain. coming to the boundary condition on the surface of the geometry, for this problem, the geometry is fixed, it is not moving, hence we specify no slip condition on the surface and all the four surface as well as span wise. Then later we will see, what is radiation boundary condition, which is posed on the other outlet condition, there is boundary condition called slip boundary condition and that is specified in x two direction, so that is boundary here as well as boundary here – top as well as on the bottom, in this plane view, x 1, x 2 view.

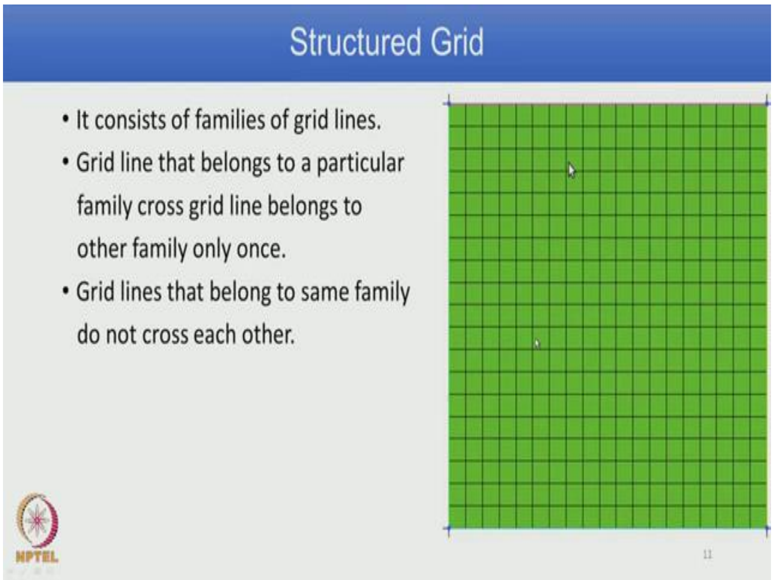
And in third direction that is x 3 direction– z direction, the geometry is actually 2D, which means there is only one and in other direction it will extend infinity. And we have taken domain size to be 4D, again there is a question how 4D is arrived. This again required some experience and looking at the problem nature. So, in this domain in the x three direction is extended infinitely, and because of that the boundary condition forced is periodic, which means whatever solution happens at this end is forced at this end. Similarly, whatever solution is obtained here on one side of the z-direction is posed on the other side of the z-direction, such a boundary condition is called periodic boundary condition. So, you observe here, if a problem is defined, there is need to specify optimally domain extend in all the direction, and specification of the boundary condition in all the boundary locations.

(Refer Slide Time: 12:12)



Here is an another example problem, the geometry is somewhere here, and the flow is from this direction and go into the other direction. And you can see here, only one quarter of the geometry is shown, this is supposed to be the domain, one quarter of the actual domain, the problem there is a symmetry, hence this is taken only one half and then you can repeat by taking a mirror image of this domain.

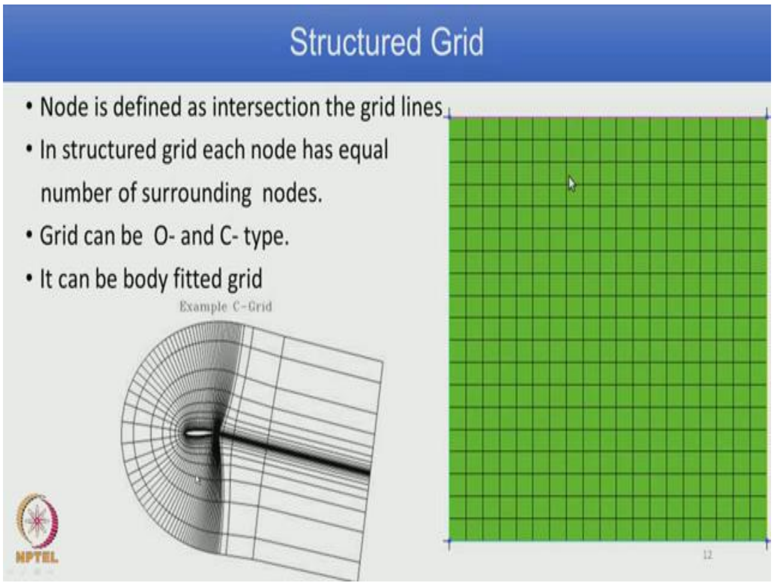
(Refer Slide Time: 12:50)



So, we go to the definition of grid, we mention once we define computational domain, we need to generate mesh including the geometry in the domain. And there are two classifications mainly; one is the structured grid, another one is the unstructured grid. We look into the definition of structured grid. So in structured grid, it consists of families of grid lines, so in this figure, that is shown here, black line that you are seeing running vertically as well as horizontally. Each one of them represent one grid line, and all the vertical grid lines are one family, and all horizontal grid lines are another type of family. So in structured grid, grid line that belongs to one particular family cross grid line that belong to the another family only once. So, if you look at this figure, horizontal lines belong to one particular family crosses the vertical line that belongs to another family at only one location. So for example, here, the same horizontal line, it may run through, but it will meet or cut or cross another vertical line at some other location.

Similarly, if you take any vertical line which belongs to one family, it crosses any other horizontal line at only one place. So, if you take this particular line, it crosses the horizontal line here then if you go along the same grid line at this location, it meets horizontal line at some other point. So, it crosses another family at only one location. Similarly, grid lines that belongs to the same family do not cross each other. So, in this case, as I mentioned before, all the vertical lines are one family, all the horizontal lines are another family. You can observe here vertical lines do not meet each other, so any vertical lines, it runs parallel to another vertical line, another vertical line, another vertical line, they do not cross each other. Similarly, if you take horizontal line that belongs to all horizontal line belongs to one family, they run parallel to each other, and they do not meet each other.

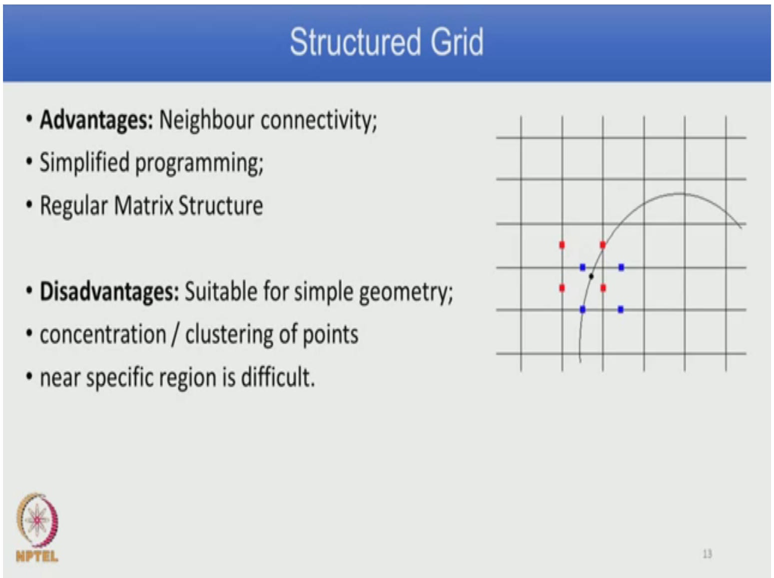
(Refer Slide Time: 15:26)



So, node is defined as intersection of grid lines. So as I mentioned before, if you take one horizontal line and one vertical line, the intersection point is actually a node. So, in structured grid, each node has same number of surrounding nodes. So, if you consider here this is particular node has surrounding nodes, say one top, another top, another node on the top, so three nodes on the immediate top, then one node on the side -right side, one node on the left side. Similarly, three nodes are there on the bottom, so 1, 2, 3 for this particular node. And you go anywhere else in the domain, so for example, if you take here then you observe that same three number of nodes, three nodes on the top, one left, one write and then three nodes on the bottom. So, in structured grid, each node has equal number of surrounding nodes. So, there are two important points to qualify or to qualitative structured grid one family line do not cross the same line, and one family line crosses another family line only once.

And depending on the overall arrangement of the grid, grid can be called either O-type or C-type or H-type. For example, in this, this is the flow past aerofoil, it is the structured grid, and if you look at the overall shape, it looks like a alphabet C, so this is what is called C-grid. And in this case, it can be body fitted, so this is an aerofoil, and you generate one family of line close to the surface. You following the surface of the geometry, so you get that lines shape similar to the geometry shape, and another family of line running perpendicular or in other direction emerging from the surface, so you are seeing here all this line almost vertical after some distance, and it is almost horizontal that is another family of line.

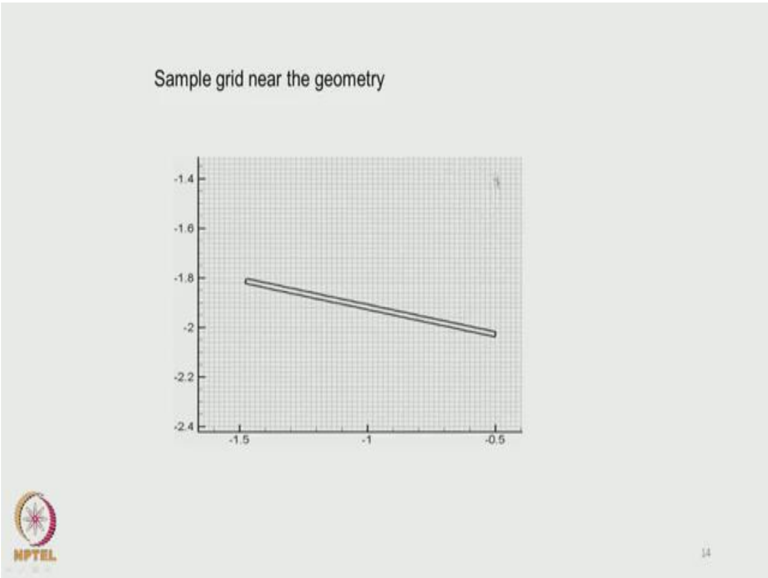
(Refer Slide Time: 18:11)



So, what is the advantage, in structured grid, as we have seen, we know for a particular node what are the surrounding nodes and we know it is the same number of surrounding nodes throughout the domain so that makes our job easier, so it is easy to have neighbor connectivity. And because of that it is easy to do programming, because we know that you can definitely say so many numbers are there, and it is the same row. And later we will see, all discretization of the governing equations will result in matrix form, in the sense A X equal to B that kind of form we will get; and A is the coefficient matrix, which usually comes from grid related quantity. And in structured grid, because of this specific behavior of equal number of grid points everywhere the matrix appears to be very regular, hence there is definite procedure or simplified procedure to invite a matrix.

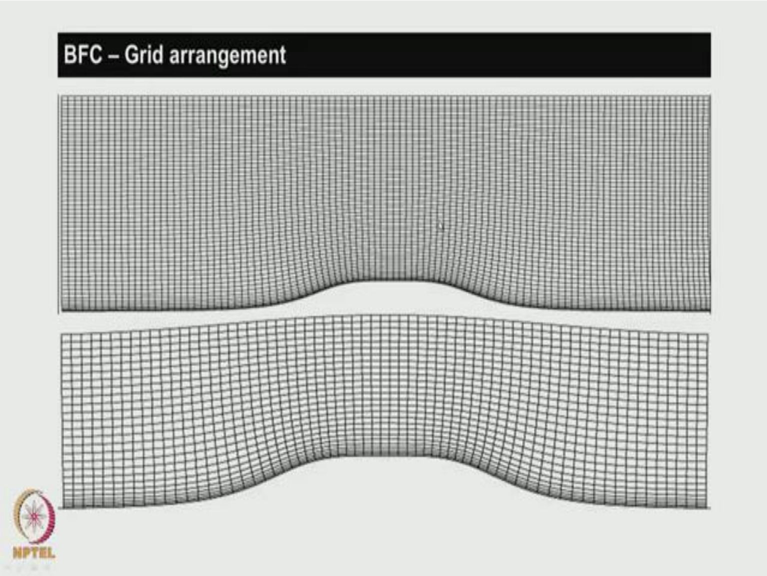
It is not only always advantages, it is also disadvantages. It is suitable for simple geometry. And there are regions where you are actually looking for a gradient or flow undergoes through gradient or there is a sudden change in the geometry where you want to have a clustering of mesh, because it is so structured, it is very difficult to have a concentration or clustering of grid points. Similarly, there are some flows, there are some region in flow, where you are looking for a specific region, for example, there is injection, there is a low pressure region, or there is a heat source then those specific regions requires grid of different type or very fine mesh or completely a different from the structured mesh, so it is very difficult to have a structured grid throughout the domain for this specific regions.

(Refer Slide Time: 20:35)



So, here is another example, this is the plate inclined and on the background you have seen structured mesh, regular rectangular mesh. And what is shown here is zoomed you, very near the geometry, and you can observe, the same uniform mesh everywhere, because you want to have a fine mesh near the geometry and that is possible to do, but elsewhere you get fine mesh.

(Refer Slide Time: 21:08)



This is another example, flow past two-dimensional hill. And this is again a body fitted grid. So as you can see the geometry is going like this, and you have one family of grid line following the geometric profile shape very closely, and it spreads as you are going in that direction. There is another family line, which is showing here vertically and you also observed that it changes slightly to satisfy a condition called normality of the surface, and this is the body fitted grid. So, for the same geometry, you are seen here two types of two mesh, both are structured, both are body fitted. The bottom one is the coarse mesh, the top one is the fine mesh for the same geometry by the same grid arrangement, but number of grids are more in the fine that is shown in the top and less that is shown in the bottom is called coarse mesh.

So, in today's class, we got the definition of domain, then we went to the definition of structured grid. I try to explain with some examples, and before that we had seen steps involved in CFD, there are three main steps, and we did some explanation on task involved in each steps. So, in next class, we will see examples and definition of unstructured grid and then we slowly move onto finite difference procedure.

Thank you.