Fluid Mechanics

Prof.T.I.Eldho

Civil Engineering

Indian Institute of Technology, Bombay

Lecture - 29

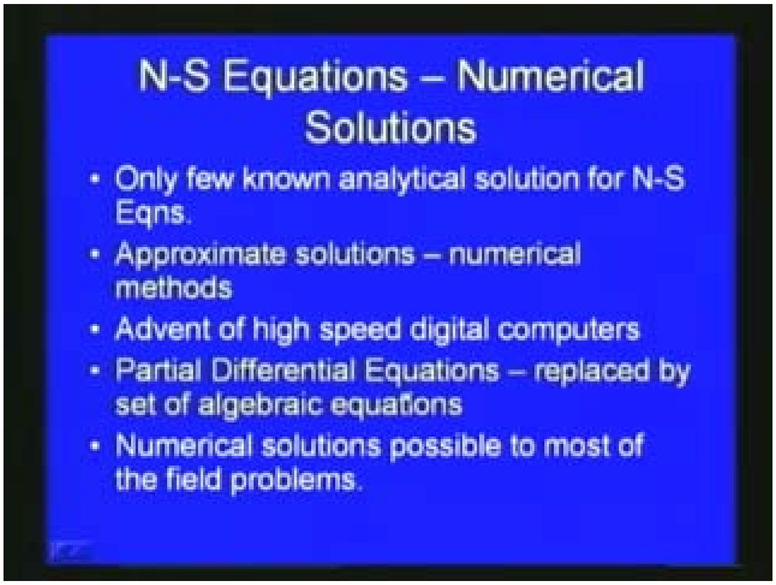
Navier-Stocks Equations and Applications

Welcome back to the video course on fluid mechanics. In the last lecture, we were discussing about the navier-stocks equations and its exact solutions. So, we have seen various cases wherever exact solutions are possible. But as we can see, only few cases wherever lot of simplifications are possible, the problem is so simple, domain is so simple, and number of assumptions, say, parallel flow like that we can put for, then only we can get exact solutions. However, most of the field problems wherever we try to solve real field problem, then we can see that these kind of assumptions if you put, then the problem becomes so simplified. Then, we will not get the results what we expect. Generally, these kinds of analytical solutions we discussed, say, most of the field problems, we cannot directly apply.

So, we have to solve the navier-stocks equations with the continuity equations in its full form. Then, we have to get various parameters like velocity pressure, vorticity, or other kinds of parameters. So, to do this, we do not have any short cut, we have to solve the navier-stocks equations, say, approximate methods or numerical methods. With the advent of the various computer technologies and various advanced numerical methodologies, we are able to solve these navier-stocks equations and then try to get solutions for various field problems. Today, we will discuss various numerical solutions for the navier-stocks equations. Then, we discuss various methodologies, numerical methods that are used briefly and then we will see how the applications of navier-stocks equations with respect to the numerical method.

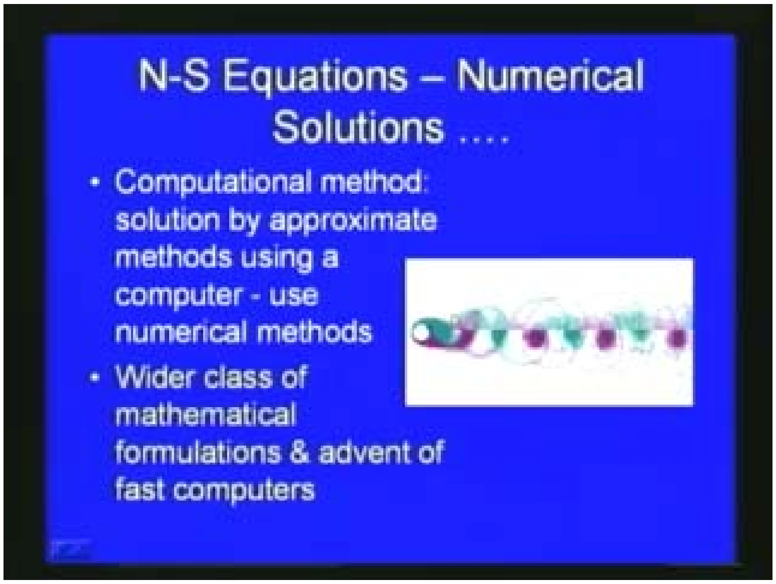
Here, we have the navier-stocks equations of the numerical solutions. So, only few analytical solutions and approximates solutions are available here. We have to go for appropriate solutions. As I mentioned, high speed digital computers have helped a lot in this kind of development. Hence, I called this as ‘computational fluid dynamics. Here, what we are generally doing is that the partial differential equations, which we say, the navier-stocks equations, which we are seen in three dimensions or two dimensions or partially refresh equations.

(Refer Slide Time: 03:40)



By using the numerical methods, we are trying to replace this partial refresh equation by a set of algebraic equations. So, once this transformation is done, then we can apply the boundary conditions into the system of equations. Then, we can get the numerical solutions. As we are trying to solve the complete navier-stocks equations this way, that means, by using the numerical methods, then these numerical solutions we can apply for the most of the field problems.

(Refer Slide Time: 04:44)

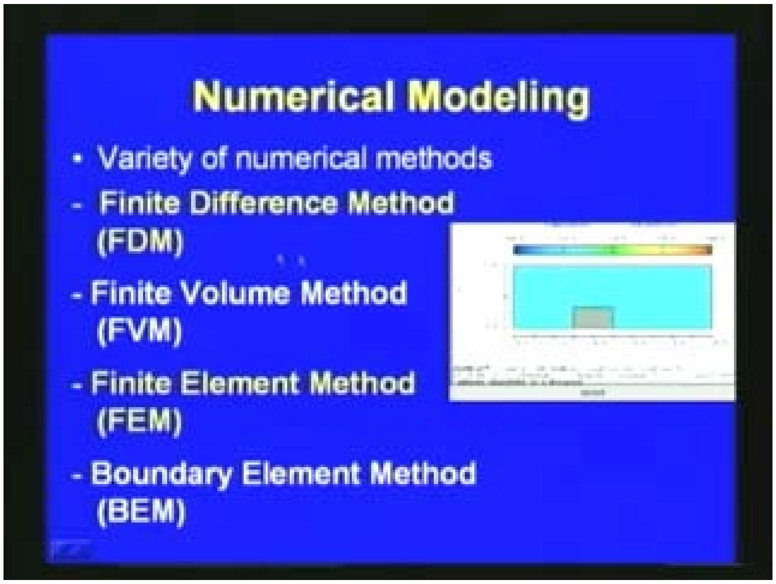


Here, it is the computational method solution by approximate methods. So, we are providing an approximate method; it may not be hundred percent right. So, various numerical methods are used. For example, when we consider the flow fast cylinder like this, you can see that there we cannot approximate these kinds of problems. We have to solve the durial navier-stocks equations and then we have to obtain the various parameters like velocity. Then, we can represent it visually like animated or we can represent in terms of graphical or other tabular forms.

So, wider classes of mathematical formulations are possible. We have already seen the navier-stocks equations, three different forms such as primitive variables, velocity-vorticity formulations, and vorticity-stream function formulations. Depending upon the type of the problem which we are trying to solve, we can formulate the problem mathematically. Then, we can solve using the numerical solutions. So, this branch of fluid mechanics is called computational fluid dynamics. The navier-stocks equations are solved using these numerical methods and then, we try to solve the real field problem.

Commonly, in literature, a number of numerical methods are available. But, generally, you can see that we use four numerical methods such as finite difference method, finite volume method, finite element method, and boundary element method.

(Refer Slide Time: 06:06)



So, these are the commonly used numerical methods. Each method has its own advantages and its own disadvantages, say various difficulties. Depending upon what kind of problem we are trying to solve, we can use any of these: finite difference method or finite element method or finite volume method or boundary element method. Then, we can try to approximate the navier-stocks equations and then try to solve.

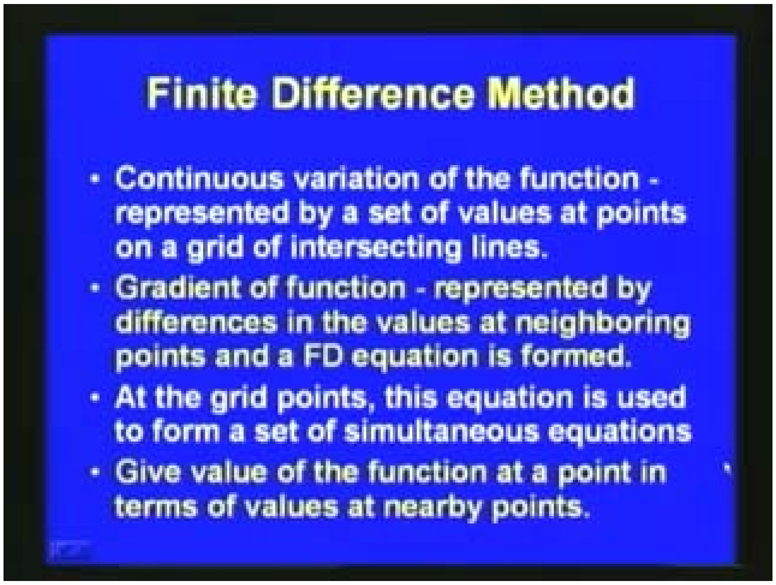
So, the selection of methodology depends upon various factors such as the type of problem which we have to solve, the familiarity of the user say using this methodology, and then the availability of the kind of computers which is required for the typical like boundary element method, we have to solve it large set of equations. So that kinds of methodology may need slightly advanced computer.

Like that, depending upon the user and depending upon the problem, we can choose any of these methodologies. For example, if you want to simulate what happens, say, here, you can see that, we want to see this is a cube, we want to see the flow behavior or flow surrounding a cube. So what we are trying to do is, we will choose one of these methodologies and then we can discretize this domain. For example, here, this is the domain, so we can discretize the domain and then, we can set approximate the navier-stocks equations. As we have already seen the partial friendly equations, we will be transformed to all the finite set of equations and then we will be applying the boundary condition.

Say, for example, if the boundary condition can be say in this figure, what is entering from here? Say, if you know the velocity at this location and here, we will be having no slip conditions and then, here vorty of the conditions. So, based upon the boundary conditions, we will be solving the problem. Now, say, we will be briefly discussing all these numerical methods like analytical methods, finite volume method, finite element method, boundary method very briefly. Since I want to discuss here is, only, say, what the methodology is and which way we are trying to solve the problem.

But, otherwise, each of the methodology, say, to describe or to explain to all the level, it will be very difficult. So, here, we will be discussing briefly. First one is the finite difference method. In finite difference method, the continuous variation of the function is represented by set of values at points on a grid of intersecting lines. So, what we are doing in finite difference method? Let us consider the flow in square gravity like this. As we have seen this is a typical problem in two d, what we are saying in finite difference method? What we do? If this is the boundary, here is gamma one, gamma two, gamma three, and gamma four. This is the domain.

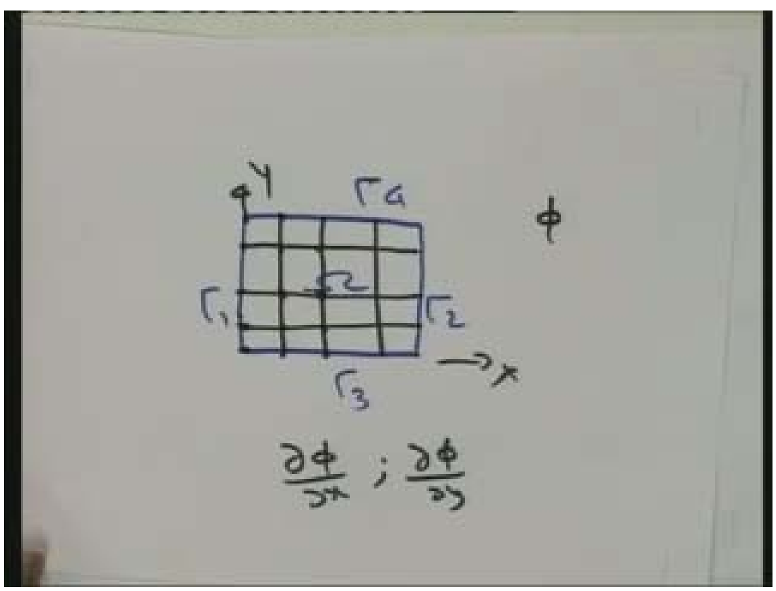
(Refer Slide Time: 08:08)



What we do in finite difference method is, we will be representing this domain with respect to a grid points. So, here, we can say, just discretize the domain to grids, that is square or rectangular grids like this. Then, we will be representing the domain with respect to grid points. So, these are the grid points here. Then, say, this is x direction and this is y direction. We can represent each grid point and then what we are doing? So, the convious various of the function, the function can be the velocity or whatever the function we are trying to approximate, we will representing the functions by a set of values at points on a grid box intersecting lines.

So, you can see a grid of intersecting lines here. Then, the gradient of the function, say, here, if phi is the function which we try to approximate here. Then, you will be having the gradient like del phi by del x or del phi by del y. So, the gradient of the function is represented by the difference between, say, the differences in the values of neighboring points. So, you can see that, say, between this point and this point, we will be trying to represent between the neighboring points. That is the way we represent in the finite difference method. Between these two points, we take or between these two points, we represent the del phi by del y.

(Refer Slide Time: 10:52)

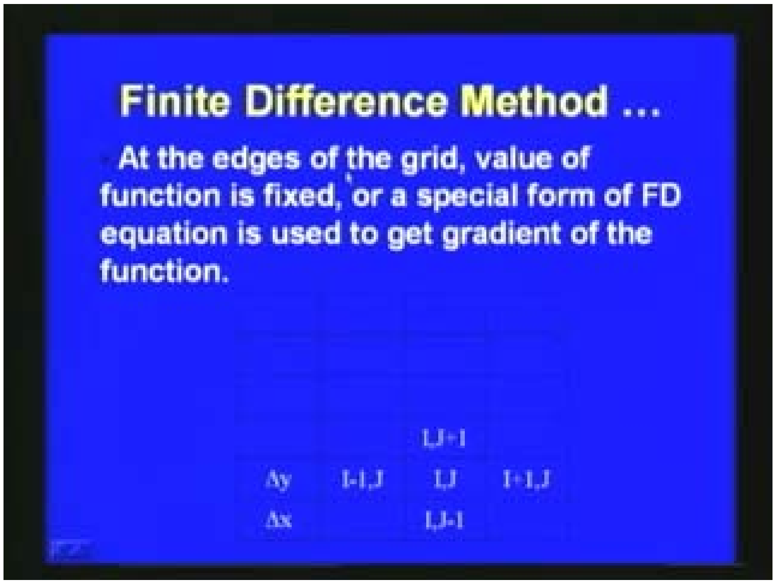


Like this, we represent the gradient of function represented by difference in the values at neighboring points and finally, finite difference equation is found. So, finally, after representing, say, after writing the equations, say, each grid point, we can write the equations. Finally, we are representing the partial differential equation, say in this case, the navier-stocks equations and continuity equations, with representing in terms of difference equation. So, we form the difference equation. At the grid points, this equation is used to form a set of simultaneous equations.

So, we have various grid points as explained here. So, this equation is used form a set of simultaneous equation. Finally, it gives value of the function at a point in terms of the values at nearby points. So, once we form the difference equation and then, we will know the boundary conditions or at least few grid points will know the values, with respect to the non–values, we will be trying to find out the unknown values at point in terms of the values at the nearby points

So, this is the procedure used in finite difference method. Here, we obtain the value of a function at a point in terms of values at nearby points. If there is any specific case at the edges of the grid, value of the function is fixed, or a special form of a finite difference equation can be used to get gradient of the function So, depending upon the problem, say, at some places, we have to say, what we have discussed here is only the basic or the fundamental, which way the finite difference method is working. So, there are number of variations of finite difference that are available in literature. Also, the variation with respect to space, with respect to time, there are number of methodologies for the finite difference method available.

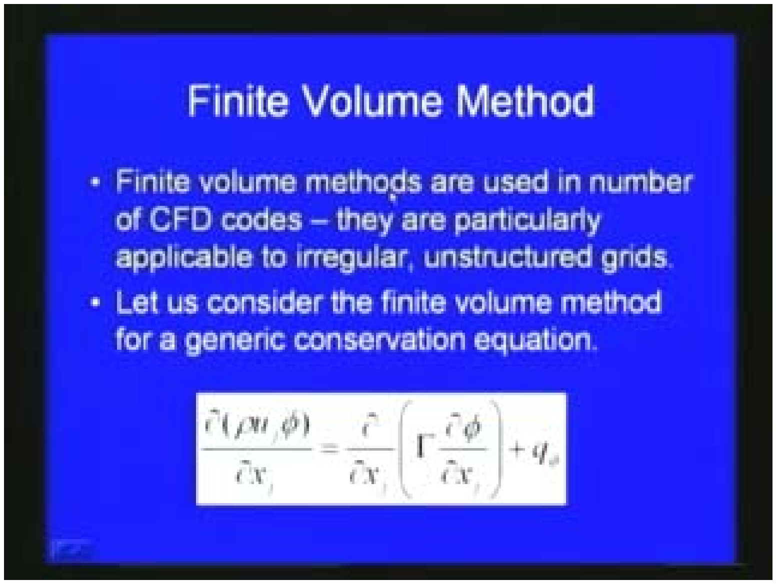
(Refer Slide Time: 13:03)



Essentially, what we are doing here is, as we are already discussed that between two grid points, we see the difference, say the gradient. Then, we try to approximate the gravany equations, say, navier-stocks equations. Then, write the difference equations and then all the independent equations with respect to the non-values or the boundary conditions. So, this is the finite difference method. Another important methodology used in computer fluid dynamics is the 'finite volume method'.

Here, in finite volume method, you can see that a large number of CFD codes are written now-a-days, since it has got number of advantages. So, these finite volume methods are particularly applicable to irregular and unstructured grids. We have seen that finite difference method, generally says we use square or rectangular type grid.

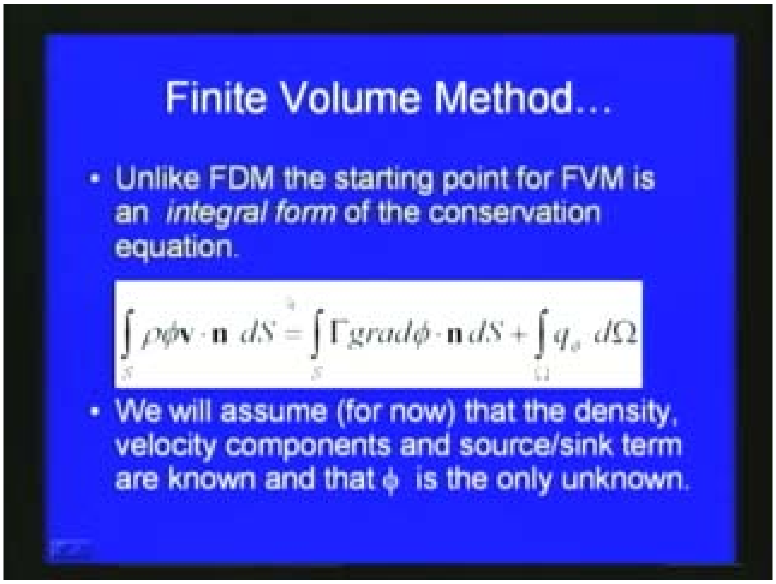
(Refer Slide Time: 13:09)



So, wherever irregular shape or unstructured grids comes, it will be more difficult to deal. Hence, finite volume method has advantages over finite difference method to deal with the irregular unstructured grid. If you consider the finite volume method for a generic conservation equation like this, same del of rho u j phi by del x j del by is equal to del by del x j gamma del phi by del x j plus q phi. Here, phi is the function, u is the velocity vector, rho is the density, and q is show sourcing. So, let us consider a typical equation like this. By using the finite volume method, here, we consider the unit cells like this. So, it can be regular cell structure like this; rectangular or square or we can have any shape of the cell. Then, in the finite volume method, the domain is discretise and then, set of nodes and grid cells. So, you can see that, the nodes can be on the intersection or it can be at the middle.

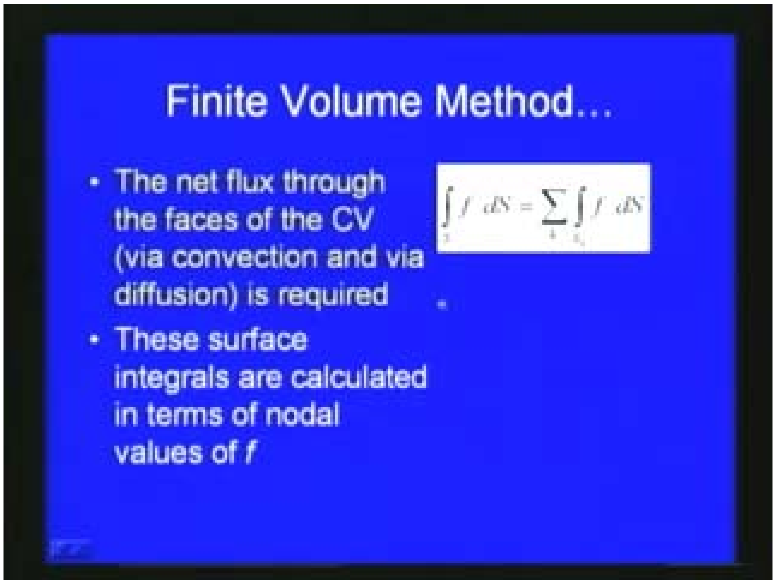
Here, you will assume that the grid nodes are located at the grid cell centers. So, this is not the only option; other way is also possible. Then, compare to the finite difference method, the starting point for finite volume method is an integral form of the conservation equation. The conservation equation, which we will write in the integral formula, in this integral, with respect to s is rho of phi v dot n ds, n is equal to integral s gamma gradient phi dot n ds plus integral omega q phi d omega.

(Refer Slide Time: 15:20)



So, we write the equation like this, which we have already considered. We are integrating with respect to the cell, which we consider. Here, we will assume that the density, velocity components, and source or sink term are known and the unknown is phi. Then, we can obtain the solution. The net flux through the faces of the control volume, via convection and via diffusion, if it is required, we can write integral s f ds is equal to sigma k integral s k f ds. So, we can integrate throughout the various cells.

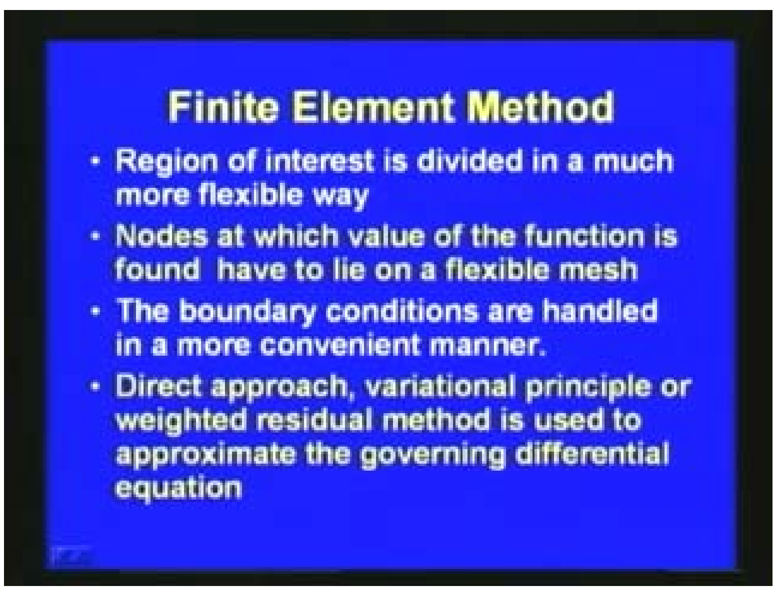
(Refer Slide Time: 15:55)



These surface integrals are calculated in terms of nodal values of f. So, this is the brief procedure as far as finite volume method is concerned. Here, compare to the finite difference method, as we have discussed earlier, say, if we consider the navier-stocks equations and any kind of irregular domain, we can use the finite volume method. Here, basically, we discretize the domain into sales and then we consider the grid point either at the center or at the intersection of the grid. Then, we approximate the equation as we have already discussed and then we integrate with respect to the cell to get a solution. So, this is very briefly, if you say, finite volume method was like this. So, we can approximate navier-stocks equation, using the finite volume method and then we can try to solve for various problems. This is the finite volume method and then another important methodology used is called 'finite element method'.

Here, the region of interest is divided in a much more flexible way. The nodes at which value of the function is found have to lie on a flexible mesh. The boundary conditions are handled in a more convenient manner. There are a number of approaches in finite element method.

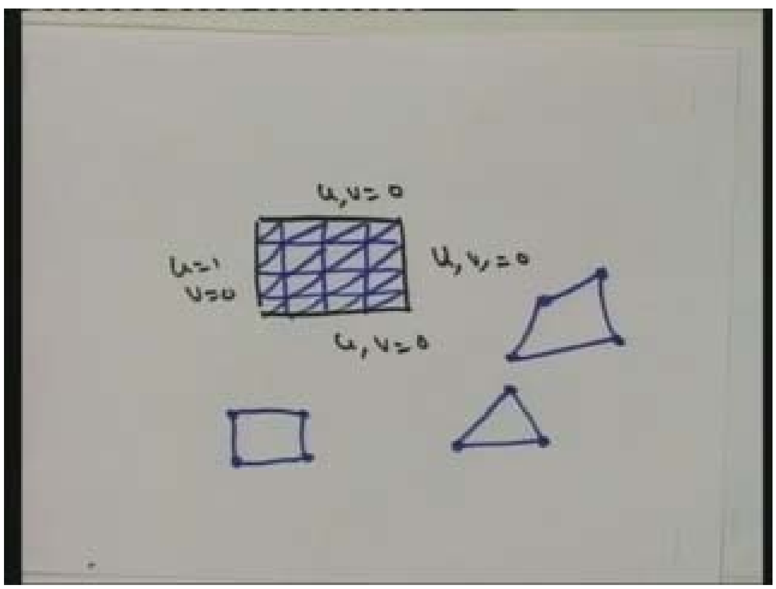
(Refer Slide Time: 17:23)



Some of the commonly used methodologies are: direct approach, variational principle, or weighted residual method, which can be used to approximate the governing differential equations. For example, if you consider the problem, which we considered earlier like the square gravity problem, using the navier-stocks equations. If you want to show, say, let us consider gravity like this. If u is equal to one, v is equal to zero; here u, v is equal to zero; u, v is equal to zero; u, v equal to zero. This is the problem and then what we do here? Compare to the finite difference method, this is much more flexible in finite element method.

So, what we can do? We can discretize the domain, say, instead of rectangle source, we can use triangular elements like this also. So, the advantage here is, methodologies are much more flexible when compared to the different kinds of elements, like we say, we can have triangles or we can have rectangle or square or we can have same quadrilateral like this.

(Refer Slide Time: 18:53)



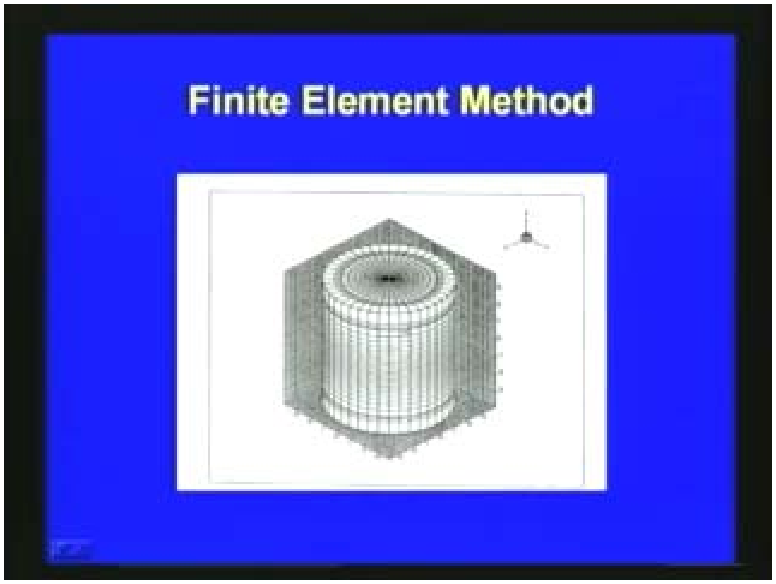
Here, this is called an element and say, we define here the nodes. So, these are called nodes and here, various shapes of elements are possible. The advantage here when compared to any other numerical methods is, here we can have the different shapes of elements. So, we can easily deal with any kind of irregular domain and then say, we can fit the domain with an accurate mesh compared to the grids, which we are using in finite difference method. Then, say, after the meshing is over, the view the same, defining an interpolation function.

That is, the variation like the velocity or the parameters, which we consider in a best stock equation either the velocity or pressure, then we will be approximating these functions or the parameters, velocities, and pressure with respect to interpolation function. Then, we will be trying to approximate the function with respect to the interpolation function. Then, we will be putting back to the relevant equation and we will try to orthonolize it. Say for example, one of the methodologies, the caliarchy finite method; we try to orthonolize with respect to the interpolation function or the shape function. Then, we force the error to zero.

Since we are approximating with respect to interpolation function, solution will not be exact. So, we are trying to force the residue produced to zero, so that the error is vanished. Then we get an approximate solution. So, as discussed in finite element method, there are number of methodologies, like direct approach. For example, when we try to solve a network of pipes, there we can directly use the dashis this back equation to get relation between various pipe elements. So, that kind of problem, we can solve using the direct approach. Another important methodology is called variation principle. So, variation principle generally, say, we have to derive a variation function for the relevant equation, which we consider and then we are trying to approximate.

So, this variation principle, generally, we use for structure mechanics problem. Then, another important methodology is the latter issue method. So, they are same. We are approximating with respect to interpolation function. While approximating, due to error, the residue will be made and that residue will be forced to zero. So, that is what we generally see in the method of weighted resolve approach, say, one of the commonly used methodologies in the problem. Like this, different methodologies are there in finite element method. For example, here, if you consider, say, in this domain, there is a cylinder and if you want to see that, how the moment of a sphere inside the cylinder is.

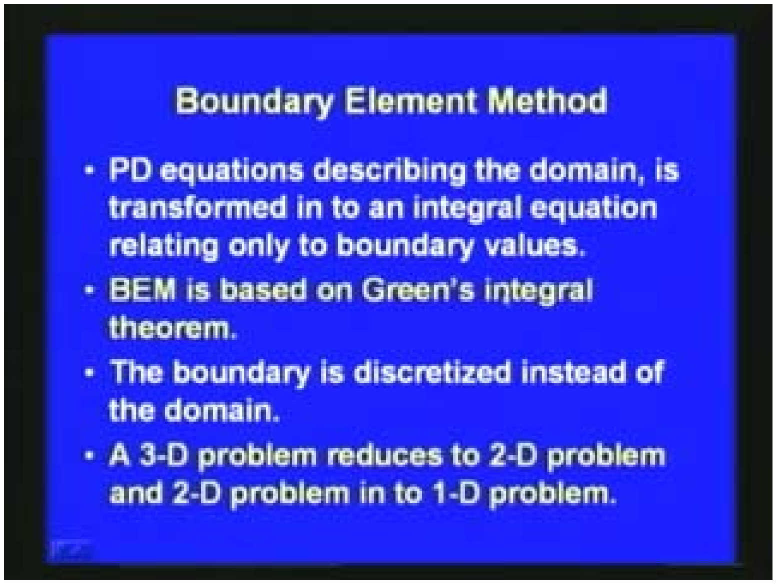
(Refer Slide Time: 22:10 min)



So, what we can do is, we can discretize the domain using three dimensional like this. Then, we can solve the navier-stocks equations three dimensional, so that, same we can initially transform the partial differential equation, navier-stocks equations into all the finite set of equations. Now, we can apply the boundary conditions and we can solve the system equations to get the unknowns. So, this is the essential principle, behind the finite element method. So, this finite element is also very much useful, say, due to its number of advantages when compared to the finite differential methods. Finite element method is also very commonly used to solve various fluid flow problems, by using the navier-stocks equations. Lastly, another methodology that has been recently developed, recently in the sense, say, in the 1970s, the development of methodology has been started and it is called boundary element method.

Here, in boundary element method, the partial differential equations, that is, partial differential equations means navier-stocks equations, describing the domain is transformed into an integral equation relating only to the boundary values.

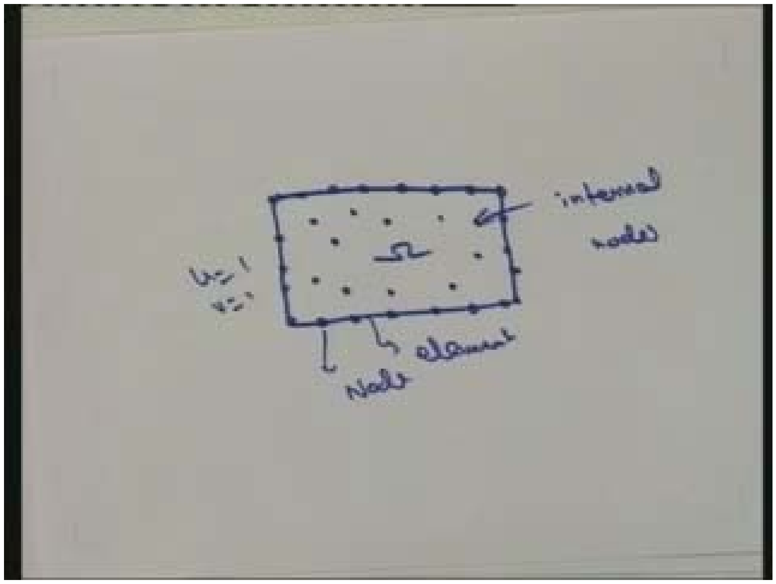
(Refer Slide Time: 23:21 min)



Here, the boundary element method is based on the Green's integral theorem. So, if you go to the advanced mathematical test, we can see that green's theorem like green's pass theorem, green's second integral theorem are there. Here, these green's theorems are used to transform the equation, which is on the domain to the boundary. So, here, we will discretize the boundary instead of the domain. Then, if you want to find out the parameters, once we can discretize the domain and we will get the equation in terms of the boundary integral. Then, we can say, for example, we can find out the unknown means beyond the boundary and then, we can find out the unknowns on the domain.

So, here, say for example, in boundary element method, if it is the domain, which we are dealing, we will be discretizing like this, using various nodes and elements. So, this is called node and this is called element. Once we find, say, for this particular problem, if the velocity here is known and some other partial is the velocities are not known, then we can find out first one, the boundary. Then, various internal points we can define separately, after the boundary internal equations are written. Now, we can solve for the unknowns inside the domain. Hence, these are internal nodes.

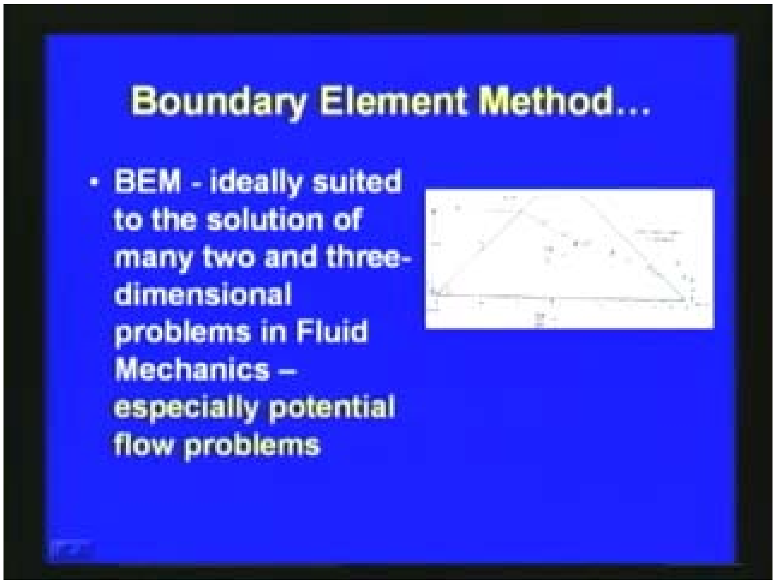
(Refer Slide Time: 25:09 min)



The advantage here is, since we are trying to solve initially on the boundary, the computational dimensions of the problem will be produced by one. So, due to this, a three dimensional problem, we can solve in two dimensions and two dimension problem computationally, we can solve in one dimension. So, this is one of the advantages of this methodology, but it has got its own limitations also. This is because, here, we have to solve the partial differential equation and then, we have to look for a fundamental solution in the boundary element method.

To derive a fundamental solution, it is very difficult for the complicated equations like navier-stocks equations, but now, recently, there are some other methodologies like dual reciprocity boundary method that has been developed. So, we can still approximate the cavern equation with respect to some other methodologies like dual reciprocity method. This is the essential of the boundary element method. So, here, same as I mentioned, say, if you want to solve flow three or dam when we discretizing like this, then we can see the element one, two, three like that, the elements and nodes we will put. Then, you try to solve. So, as for as the fluid flow problems are concerned, BEM is ideally suited to the solution of many two and three- dimensional problems especially potential flow problem.

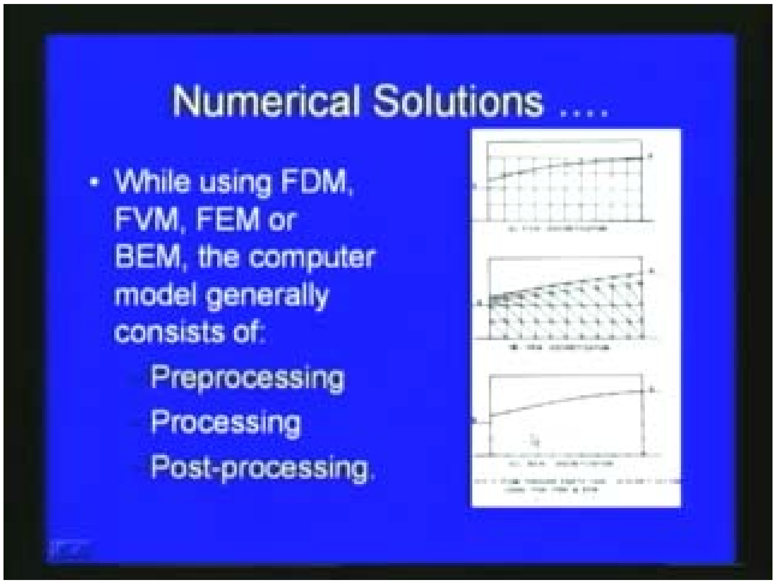
(Refer Slide Time: 26:14 min)



If you solve the fluid navier-stocks equations, these methodologies have some limitations, but the advanced techniques like dual reciprocity method can be used. Otherwise, the methodology, BEM is much more used to solve potential flow problem. So, in comparison, while using finite difference method, finite volume or finite element method or boundary element method, we generally model same like this. If you are considering flow through dam like this, then this finite difference discretization is shown here, corresponding finite element discretization is shown here, and boundary element discretization is shown here. As I mentioned, depending upon the problem, depending upon the familiarity of the user with the methodology, we can choose the methodology and then we can try to solve

So, using any of these numerical methods, there are three steps. First one is the preprocessing,

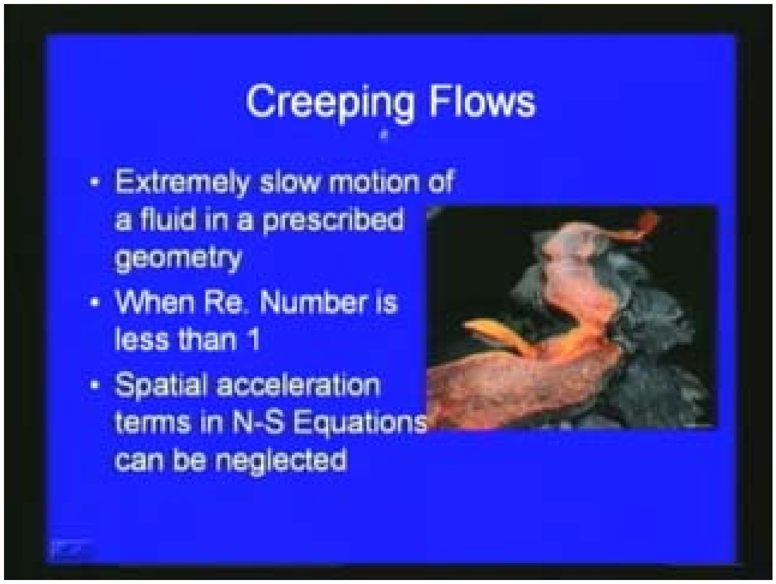
(Refer Slide Time: 26:41 min)



Here, preprocessing means, first, we will be discretizing the domain by putting the grids or mesh like this. Then, we will be putting the boundary conditions and then various parameters, which are generally available for the problem considered. The second step is the processing, which means the computer com which we will be writing for the methodology. Then, we have to run the code to generate the solution for the particular problem. The third step is the post–processing, which means once the results are generated, we will be getting in the numbers. That numbers, we have to put it in the graphical forms or the tabular form, so that the other people can understand, which way the solution has been generated and how the results are generated. So, essentially, we have three steps: one is preprocessing; second one is processing; and third one is post–processing. So, that is about the numerical methods to solve the navier-stocks equations for various problems by using different methodologies. Before closing this chapter, we will be discussing two more same typical problems, where we can approximate the navier-stocks equations and then try to get analytical solution.

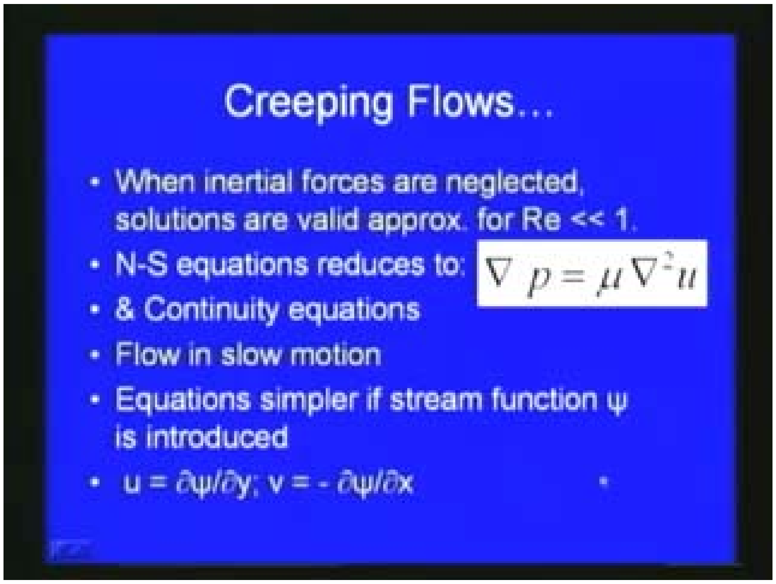
First one is the creeping flows. Actually, the second one also, which we are discussing is the random creeping flow. This is also we can classify as creeping flow, as a sub-section of this creeping flow. Here, the creeping flow is the flow, which is extremely slow motion of the fluid in a prescribed geometry. So, here, generally, by Reynolds number, it will be very low. So, it should be generally less than one. Here, you can see that this is the flow into domain, say, the lava flow from a volcano.

(Refer Slide Time: 28:56 min)



So, you can see that to be the viscosity so high and then the motion will be slow. Also, you can see that we can classify these kinds of flow as creeping flow. So, here, the spatial acceleration terms in navier-stocks equations, we can neglect, since the Reynolds number is very low, that is, less than one. So, if you neglect the spatial acceleration terms, then we get a simplified form of the navier-stocks equations.

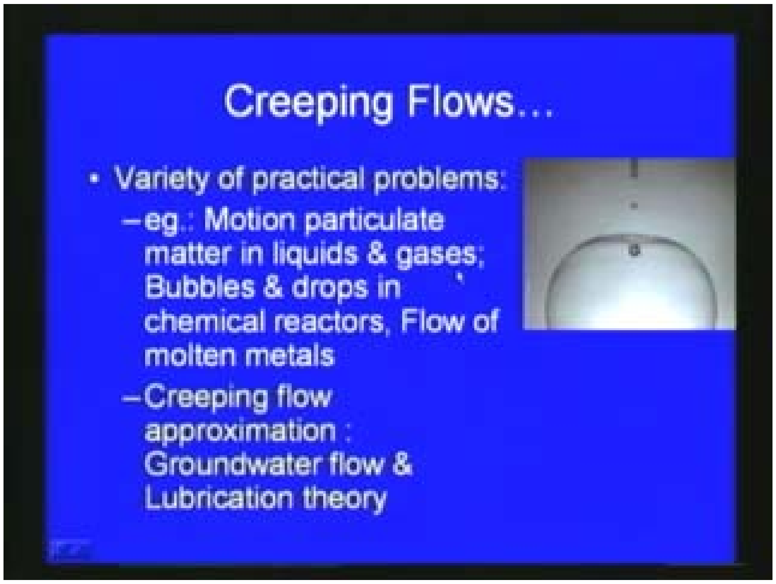
(Refer Slide Time: 29:37 min)



Generally, for the creeping flow, we can represent the equation as del p is equal to mew del square u, where p is the pressure, mew is the co-efficient dynamics viscosity, and u is the velocity. Here, when the inertial forces are neglected, generally the solutions are valid for approximately, for Reynolds number less than one and navier-stocks equations reduced to this form. Then, this form of the navier-stocks equations and the continuity equations, we will be using for the solution of these kinds of creeping flow problems. So, as we are discussing the flow is in slow motion, arbitrary high viscosity or depending upon various other conditions or situations, the velocity is very low, that is, in slow motion. Here, if you use the stream function in terms of head or the velocity, the equations will be simpler, if the function, psi is introduced.

So, we can represent u is equal to del psi by del y and v is equal to del psi by del x, and so on. Then, we can rewrite the cavern equations starting from the navier-stocks equations and then, after rewriting, we can try to solve the problems, like, what we discussed in the case of creeping flows. So, a number of problems can be solved in this category of creeping flows. A variety of practical problems like motion of particulate matter in liquids and gasses, say, just like if you consider the reservoir, the sediments will be trying to settling, since the water is stored.

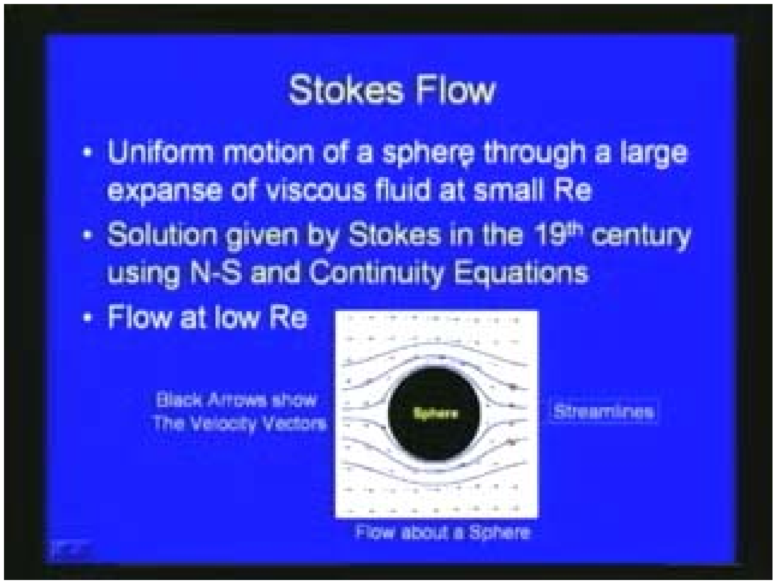
(Refer Slide Time: 31:12 min)



So, the settlement of sediments means, we can consider as a very slow motion. Then, try to get a solution for these kinds of problems or the settlement of dust particles in the atmosphere or the simulation of mist in the atmosphere or bubbles and drops in chemical reactors or flow of molten method. So, all these problems comes under the category of the creeping flows. Also, as I mentioned, this groundwater flow and lubrication problems also we can classify as a section of the creeping flow; we can approximate as a creeping flow. So, these are some of the applications of the creeping flows.

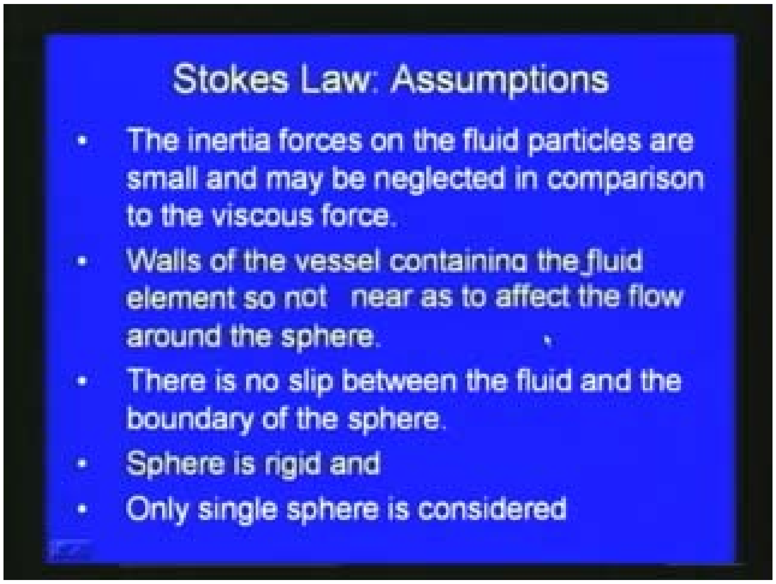
Here, we will consider two cases: one is the stocks flow and second one is the groundwater flow. So, first case is stocks flow, here, say, by using this creep flow theory, stocks try to solve by starting from a navier-stocks equations, then approximated the navier-stocks equations in the simplified form, and then we got a solution by end of the nineteen century itself. Here, stokes flow deals with the uniform motion of a sphere through a large expanse of viscous fluid is at small Reynolds number.

(Refer Slide Time: 32:47 min)



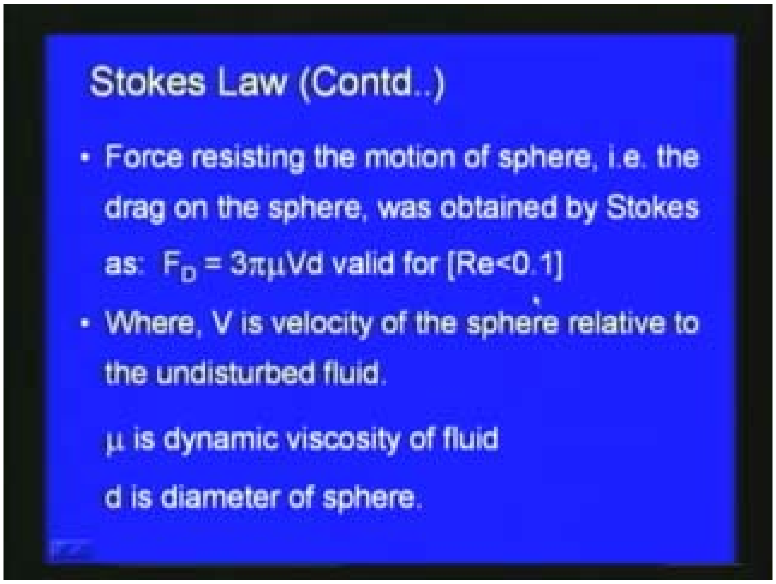
So, stokes flow deals with say, as I mentioned, these can be used in the case of sediment settlement reserve or while dealing with dust particle in atmosphere, we can use this theory, that is, stokes flow theory. This solution has have been derived by stokes by the end of the nineteen century. The flow is at low Reynolds number. Here, we can consider this as a single particle, say, as a sphere here and this shows the stream lines. Velocity vectors are also indicated. So, stokes consider the motion of the sphere through a large expanse of the viscous fluid at small Reynolds number. While deriving the solution, we put forward a number of assumptions, so that the problem will be simpler and then we get solution for the problem.

(Refer Slide Time: 33:55 min)



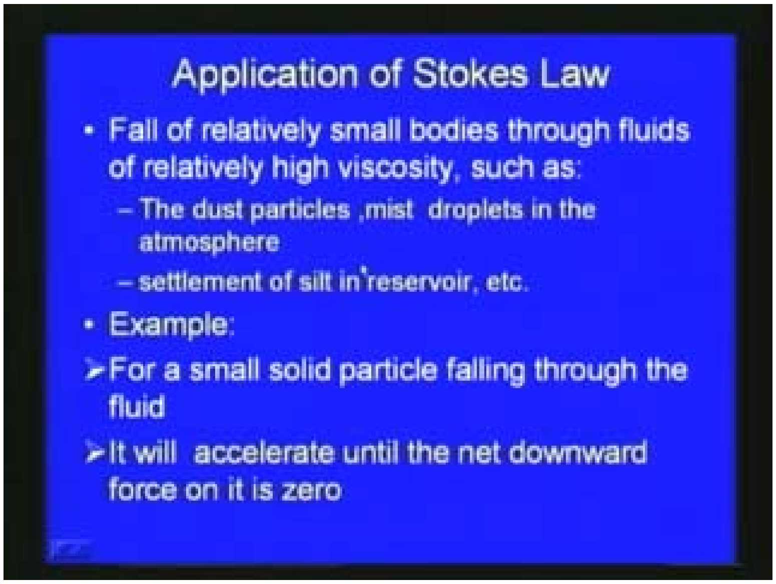
Here, the assumptions used are: the inertia forces on the fluid particles are small and may be neglected in comparison to the viscous force. Then, the second assumption is walls of the vessel containing the fluid element is not near, so that the effect of the walls are not there, when we derive the equations. Also, some of the assumptions like no slip between the fluid and boundary of the sphere and also that the sphere which we considered is rigid. To derive a solution, we used only a single sphere. So, these are some of the assumptions used to derive this solution called ‘stokes solution’. So, here, if you consider the force resisting the motion of the sphere, that is, the drag on the sphere, the stokes obtained the drag force F_d is equal to 3 pie mew into V into d. Here, we should note that Reynolds number that stokes assumed is less than 0.1.

(Refer Slide Time: 34:49 min)



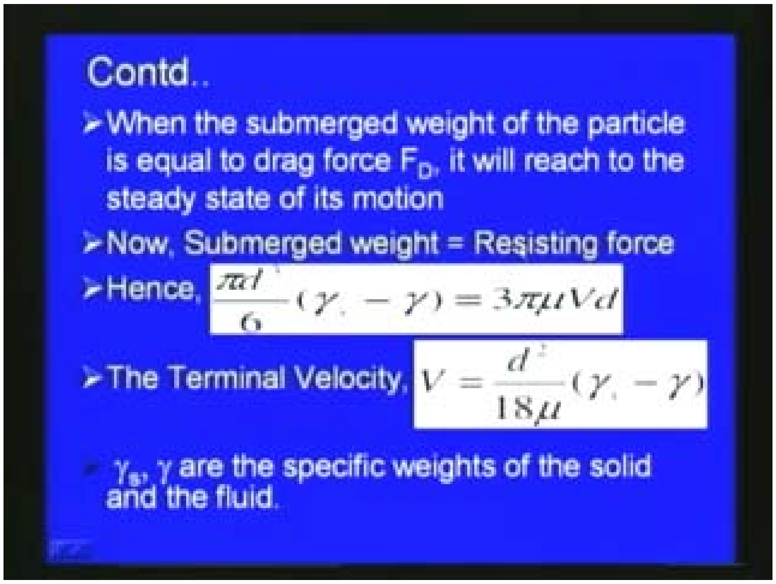
Here, V is the velocity of the sphere relative to the undisturbed fluid; mew is the dynamic viscosity of the fluid; and d is the diameter of the sphere. So, we derived that the drag force is equal to 3 pie mew into V into d.

(Refer Slide Time: 35:16 min)



For the fall of relatively small bodies through the fluids of relatively high viscosity like the dust particle mist droplets in the atmosphere and settlement of silt in reservoir, and so on, we can consider the same. For example, if you consider for a small solid particle falling through the fluid, it will accelerate until the net downward force on it is zero. So, if you consider, say, as I mentioned, either dust particle or the settlement particle in the reservoir, it will be keep on falling and the net downward force on it is zero. Then, we can use this drag force derived by each stokes and when the submerged weight of the particle is equal to drag force F_D, it will reach to the steady state of its motion.

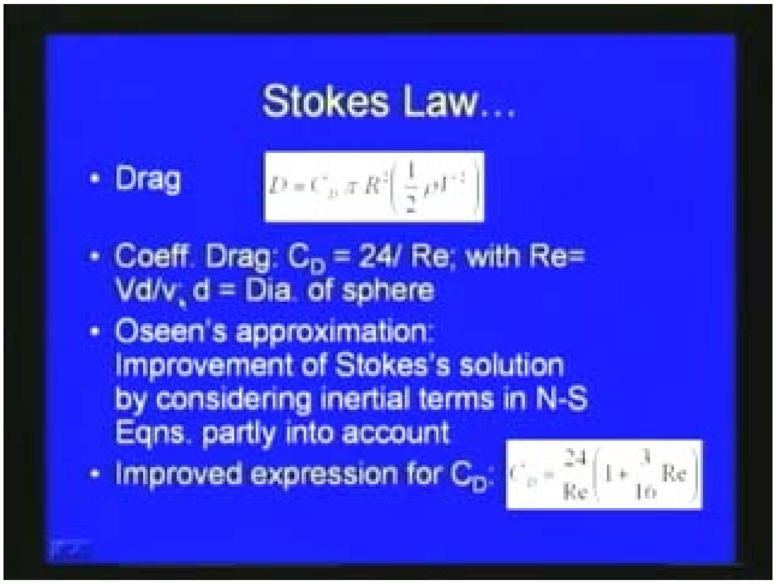
(Refer Slide Time: 35:57 min)



So, for the stokes drag force equation like this, submerged weight is equal to resisting force. Here, if we consider the single sphere of diameter d, we can write pie d cube by six into gamma_s minus gamma is equal to 3 pie mew V into d, where gamma_s is the specific weight of the solid and gamma is the specific weight of the fluid, from which we can get the terminal velocity. That means, when the submerged weight of the particle is equal to drag force, it will reach a steady state of its motion and that velocity is the terminal velocity.

So stoke over the terminal velocity as V is equal to d square by 18 mew into gamma_s minus gamma. This solution is called 'stokes solution' or 'stokes flow'. This has got application in number of fields as I mentioned, like say, reservoir sedimentation or dust particle analysis and in number of applications, we can use this stokes flow. So, this has been derived by stokes in nineteenth century. Here also, we can see that he approximated the navier-stocks equations and simplified the equations, such that very low Reynolds number problem, we can apply as derived by the stokes. Finally, an expression for drag can be written as D is equal to C_D into pie R square into one by two rho V square, where R is the radius of the sphere which we considered.

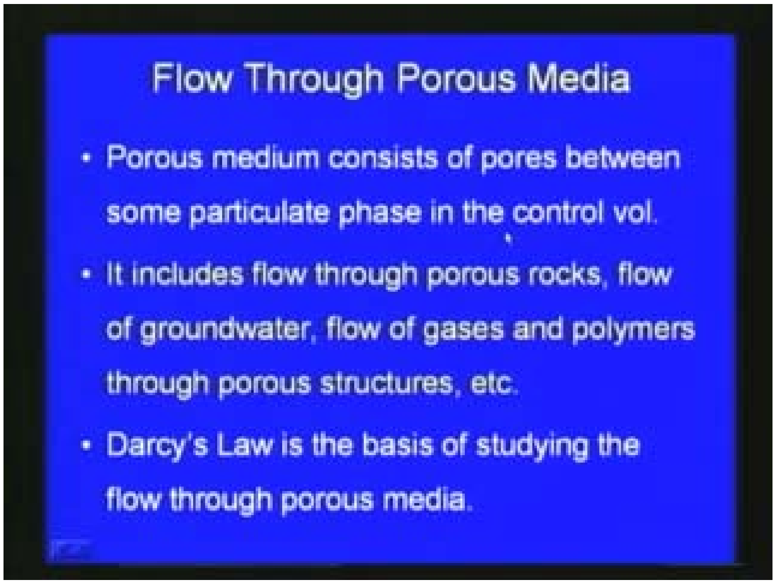
(Refer Slide Time: 37:46 min)



So, from this expression, co-efficient of drag has been derived by stokes as C_D is equal to 24 by Re, where Re is the Reynolds number. Reynolds number (Re) is equal to Vd by v; where V is the velocity, d is the diameter, v is the kinematical viscosity, and d is the diameter of the sphere. Like this, we can get the co-efficient of drag and then, later Oseen solved this again approximation. He obtained the improvement of stokes's solution by considering some of the inertial terms in the navier-stocks equations. Then, he got the co-efficient of drag as 24 by Re into 1 plus 3 by 16 Re.

So, oseen's try to improve stokes flow or stokes equations. Here, as we have seen that this problem has got number of applications, but it is a simplified form of the navier-stocks equations. Then, stokes derive the equations. Depending upon the problem, we can say, for the particular case, which we consider, we can use this stokes flow. It can be used whenever the Reynolds number is very low and then to determine the terminal velocity for the fluid particle, which we consider. So, the second problem which we want to discuss here is, the flow through porous media. The porous medium consists of pores between some particulate phase in the control volume.

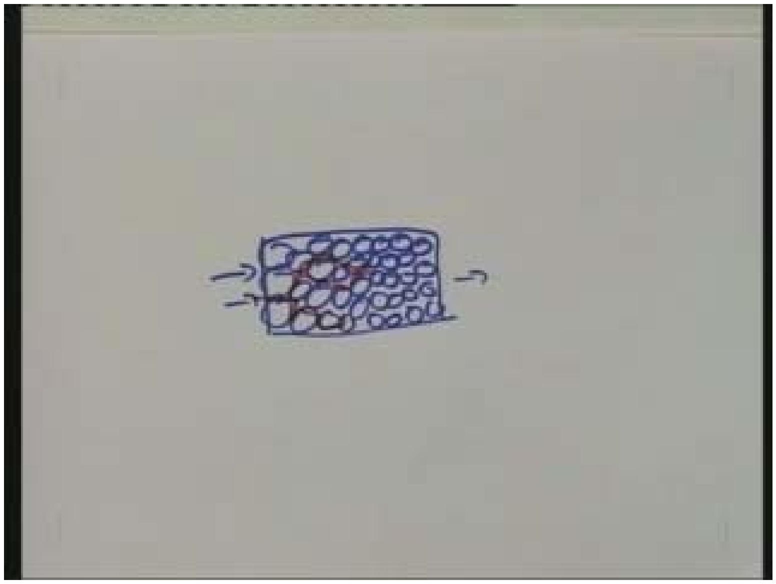
(Refer Slide Time: 39:07 min)



So, this is another important section of the fluid mechanics called groundwater flow and contaminant transport problem. Here, say, if you have got a domain like this and then there are number of particles like this. Then, we can consider the navier-stocks equations for these kinds of flow problems. Here also, you can see that the velocity will be so low, so that we can approximate, if the flow is coming like this and going through this. Then, you can see that the flow is taking through these pores only and then on the particles like this sand particles, then the velocity will be much lower through this force or of the medium. Here, we can use the navier-stocks equations and approximate creeping flow and then we can use the equation.

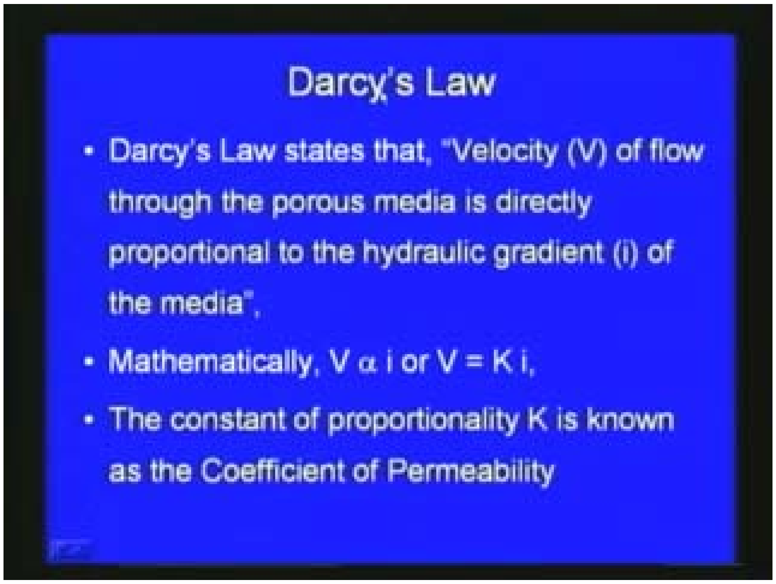
Here, we can approximate it. So, generally, say the basic equation for porous media flow is the Darcy's law. So, here, what we are trying to do is said, whatever the theory for the laminar flow through pipe, we can try to apply for this kinds of problem. So, you can see that whenever we consider medium like this, here, between the sand gradient, you can see that there is the flow that is taking place like this. So, the flow will be going like this through the different pores. What we do here? We try to approximate this problem as a pie flow that means, between the solids we consider same pipe flow, then we compare with the Darcy's and we try to obtain the Darcy's law from this theory.

(Refer Slide Time: 40:56 min)



So, Darcy's has done this experiment through the porous medium. He has derived and he has shown that the velocity of flow is proportional to hydraulic gradient. Here, the Darcy's law is the basis for the study of this porous medium flow. So, it has got number of applications, like flow of groundwater, flow of gasses, and polymers through porous structures, and so on. A number of applications are there. So, here, first, we are trying to correlate with respect to the laminar equation, which we derived for pipe, with respect to this typical porous media flow.

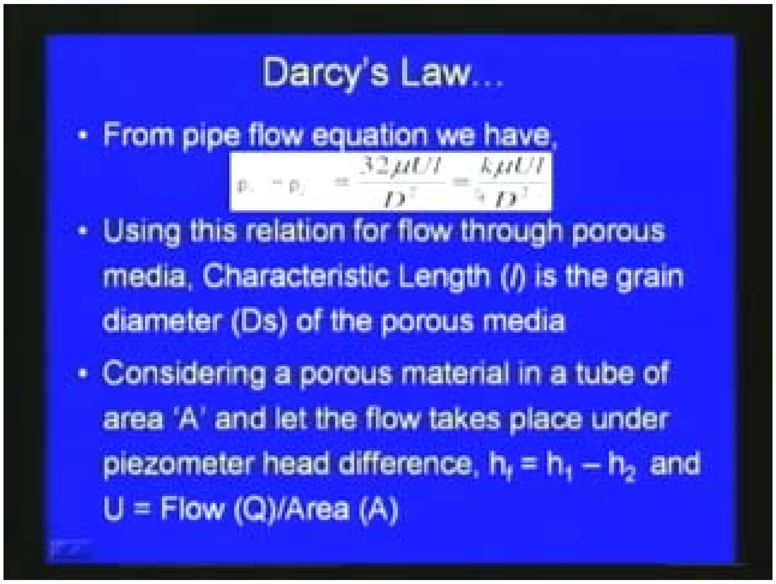
(Refer Slide Time: 41:36 min)



So, the Darcy's law as I mentioned, it states that velocity of flow through the porous media is directly proportional to the hydraulic gradient of the media. Mathematically, we can write V is proportional to i, where i is the hydraulic gradient or we can write V is equal to K into i, where K is known as co-efficient of permeability. Here, if you consider the porous media, which we are discussing as a simple case of pie flow equation, which we are derived for laminar case. We have already seen earlier p_1 minus p_2 is equal to 32 mew Ul by D square, where mew is the co-efficient dynamics viscosity, U is the average velocity through the pipe, l is the length of the pipe, d is the diameter, p_1 is the pressure at section one, and p_2 is the pressure at section two.

We can write this as p_1 minus p_2 is equal to k into mew into Ul by D square. If you assume what is happening, it is between the solid grains or the sand grains, if you consider the flow taking place the porous media flow as a pipe flow. So, we can consider the relation of flow through porous media with characteristic length as grain diameter of the porous media. Here, we consider the characteristic length (l) as the grain diameter of the porous media.

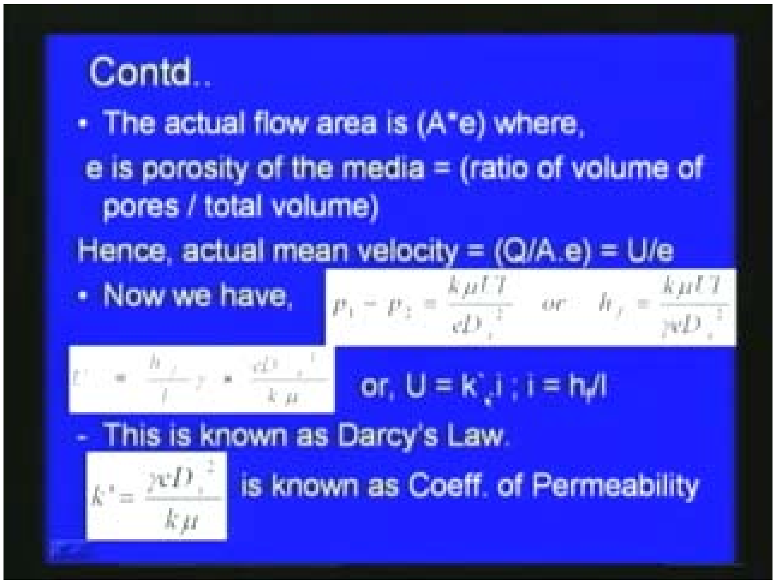
(Refer Slide Time: 42:31 min)



Considering the porous material in a tube of area 'A' and let the flow takes place under the piezometer head difference, h_f is equal to h_1 minus h_2 and the velocity U is equal to flow (Q) divided by area (A). Then, we can put in this relationship as the actual flow area. Here, we are trying to correlate with respect to the pipe flow equation and the porous media flow. However, the porous media flow that is concerned, you can see that, since here you can see various number of porous, say sand grains are there between the pores and the flow is taking only between the pores. So, the actual area will be reduced and we can write the actual flow area as A into e.

Here, A is the cross-section area and e is the porosity of the media. e is the porosity, which is the ratio volume of pores to the total volume. So, e is defined as the porosity of the media. So, the actual area is A into e; hence actual mean velocity is equal to Q by A into e and that is equal to U by e, since Q by A is U.

(Refer Slide Time: 43:38 min)



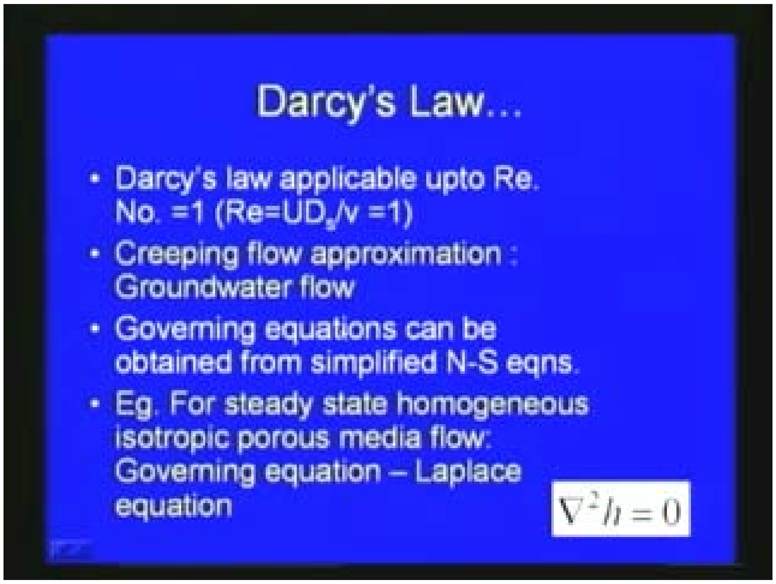
Whatever we consider the mean velocity that is used, that we have to divide it by e, the porosity. So the actual mean velocity is equal to U by e. Now, we can rewrite the equation, which we have written here as p_1 minus p_2 is equal to k into mew into U into l by e into D_s square. Here, D_s is defined as the grain diameter of the porous media or we can divide by gamma and write it as h_f, which is equal to p_1 minus p_2 by gamma. So, h_f is equal to k mew into U l by gamma e D_s square.

Finally, from this expression, we can write the velocity U is equal to h_f by l, which is the head lose, into gamma into e into D_s square by k into mew or from this, we can write as h_f by l is the hydraulic gradient, that means, the head loss. With respect to the head loss, U is equal to h_f by l into gamma into e into D_s square by k mew or we can write U is equal to k dash into i, where i is the hydraulic gradient and i is equal to h_f by l. So, this expression mew is equal to k dash i, which is known as Darcy's law. Here, k dash is equal to gamma into D_s square by k into mew as for this expression. So, this is called the co-efficient of permeability.

Here, what we did is, we try to approximate. So, you tried to use the equation, which we derived for the pipe flow and then between the sand grains, we tried to get the actual velocity and then we tried to obtain the velocity. That is the Darcy's Law. That is what is proved by Darcy through the experiment. So, you can see that this darcy's law is applicable upto Reynolds number, generally equal to one, where Reynolds number is here is defined as U D_s by new, where D_s is the grain diameter and new is the dynamitic co-efficient viscosity

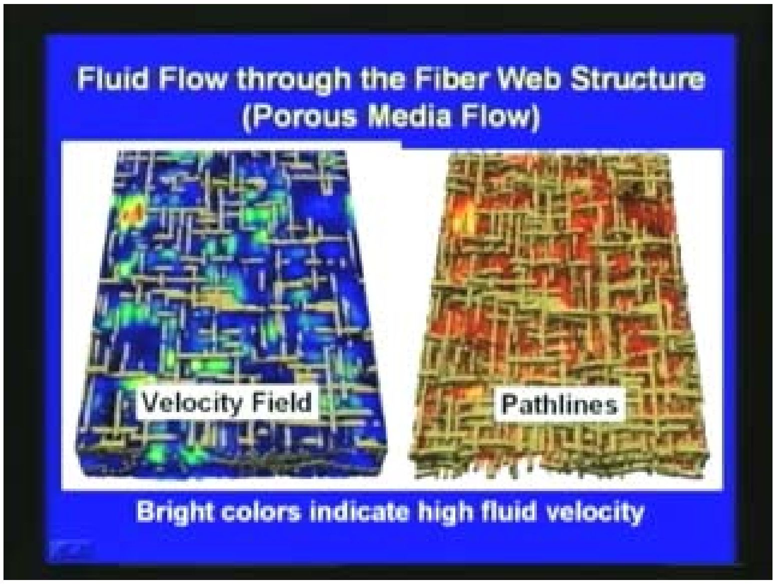
As we have seen here, this groundwater flow is an approximation of creeping flow theory which we are seen. Here also, the governing equations we can start from the navier-stocks equations and then put forward the various assumptions. Then, we can simplify the equations and then we can try to get solution. Through the simplification of the navier-stocks equations, say for example, if you consider steady state porous media flow in the homogenous isotropic porous media, the governing equation is Laplace equation. We can write from the navier-stocks equations

(Refer Slide Time: 46:43 min)



We can derive this form of the equation, del square h is equal to zero, which is the Laplace equation. Here, h is the head of the water table height for the problem considered. Like this, we can also start from the navier-stocks equations and then try to get solution. So, as I mentioned, this porous media flow is also very complex type of flow problem. So, here also, we can use the navier-stocks equations. As this slide show, say, for example, here this slide shows the flow through the fiber web structure, which can be considered as the porous media flow.

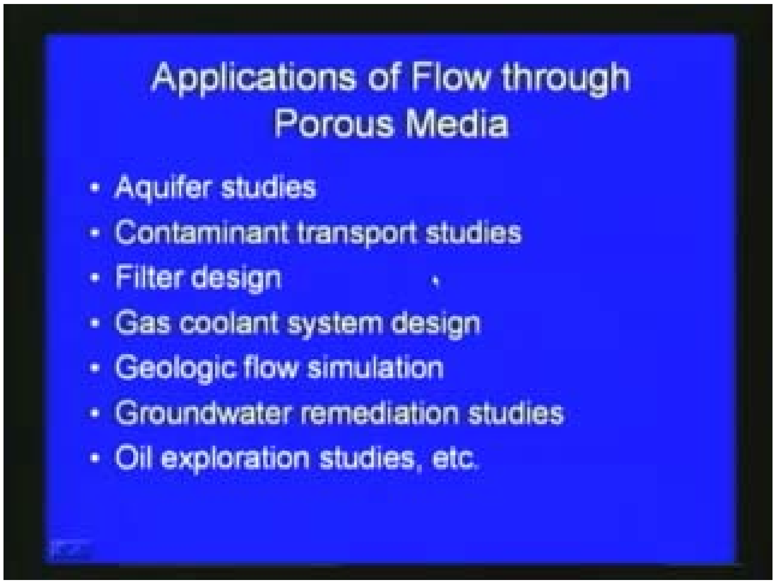
(Refer Slide Time: 47:43 min)



Here, the velocity field is also marked and then we can see that here also, path lines are also marked. So, the porous media flow, as we can see here is so complex, but still we can use the navier-stocks equations and then approximate. Then, we can try to get solution for simplified cases like flow through steady state flow through homogenous isotropic media

So, we can try to get solutions. Hence, the flow through porous media also has got number of applications like we have already seen in the aquifer studies, contaminant transport studies, filter design, gas coolant system design, geologic flow simulation, groundwater remediation studies, oil exploration studies, and so on.

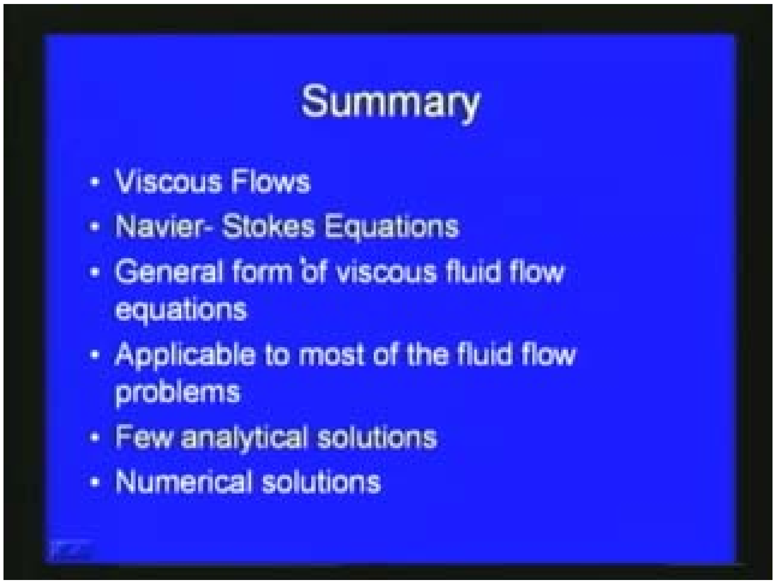
(Refer Slide Time: 48:14 min)



A number of applications are there for porous media flow. So, this is actually, I say, related to the groundwater flow, where ______ some groundwater flow can be discussed in detail. However, the aim of this discussion is stoke of the navier-stocks equations and also we can get this simplified equation for the porous media flow. Then for simple cases like same the steady state flow through homogenous isotropic media, we can try to get solution. So, like this, various kinds of flow problems can be attempted and can be solved, using the navier-stocks equations.

So, to summarize this chapter, here, we have seen the navier-stocks equations, derivation, exact solution, and numerical solutions. To summarize, most of the flow problems which we considered in practical cases or real life is viscous flow. So, we have to consider the viscosity nature; it can be newtroni or non-neutrony, depending upon the problem. So, for neutronian problem or these viscous flow problems, we have derived the navier-stocks equations. So, this navier-stocks equations is a general form of the viscous fluid flow equations. We can solve most of the problems with respect to the navier-stocks equations including the turbarian flow problems or even the compressible fluid flow by appropriately modifying the governing equations.

(Refer Slide Time: 49:31 min)



So, these equations are applicable to most of the fluid flow problems, but due to the nonlinear and second ordinates of the equations only, few analytical solutions are available. But, most of the real field or live problems, we cannot use the analytical solutions. So, to use the navier-stocks equations, we have to go for numerical methods and numerical solutions as we discussed. So, these numerical methods or numerical solutions of the navier-stocks equations, we generally call as 'Computer Fluid Dynamics (CFD). We can solve most of the problems by using the navier-stocks equations. So, this is the end of this navier-stocks equations chapter. In the next section, we will be discussing about the boundary layer flow and its governing equations and various solutions.