Computational Fluid Dynamics
Prof. SreenivasJayanti
Department of Chemical Engineering
Indian Institute of Technology, Madras

Module No.# 01

Lecture No: #01

Motivation for CFD

Introduction to the CFD approach

Welcome to the course on computational fluid dynamics. Many of you in your undergraduate and postgraduate days would have taken a course on fluid mechanics or momentum transfer; you would also have taken a course oncomputational techniques, where you would havelearnt how to solveequations;for example, a set ofand algebraic equations, or a matrix equation type ofproblems, and root finding of non-linear equations, partial differential equations, ordinary differential equations and soon.

This course, computational fluid dynamics, it derives both from fluid mechanics and also from computational techniques andcomes up with a set ofnumerical methods which will enable us to solve the equations, which govern the flow of a fluid in any domain; this enables us to go much beyond what we can do inanalytical part of a fluid mechanics course; in fluid mechanics, we have studied how flow goes through a straight pipe and how flowflowgoes over, for example, a sphere under creeping flow conditions and what type of velocity profile we have for flow over a flat plate; in such simple geometries, we know how the flow takes place and we can characterize the pressure drop or the drag coefficient or the friction factors in such simple cases; but we know that when we want to deal with practical problems, we have a difficulty in extending what we have learnt in a basic fluid mechanics courseto complicated geometries and the complicated geometries need not be very complicated.

For example, if you have a room like this, and as an engineer you want to putan air conditioner. So, the question would bethat, you would pose yourself is that where would I put the air conditioner in order to have maximum effectiveness. And so, if you want to answer this type of question, you have to know how the cold air from the air conditioner would flow in thisroom andwhat sort of coldness it would give to people occupying people in different places; you would have to understand in such a case, the buoyancy related flow which is induced by the cold air flowing from the air conditioner; and that is a complicated problem that you cannot deal with from what you would have known and learnt ina basic course in fluid mechanics.

If you take another example like, you have a chemical reactor,you take the simplest shapea cylindrical vessel which is half filled withone reactant liquid reactant and another liquid reactant is being poured into it and the two have to,let us say that, there is a density difference between the two. So, if you leave the two reactants like that, the lighter fluid would go to the top, and the colderthe heavier liquid fluid will go to the bottom; you want to have these two mixed together in order to have a reaction which we wantto take place in this reactor. So, what could we do, we could put an impeller to an agitator to mix the two fluids; the question would be, what should be the type of mixer? What should be the type of agitator? Where should we place it? And you also may want to put a catalyst for this reaction tobe catalyzedto be enhanced; andin such a case, you would like to have the catalyst thoroughly mixed. So, you may want to..,you are not only interested in how the two immiscible liquidscome together, but you also like to know how the catalyst, which may be in a solid form,would be dispersed withinthe liquid media.

So, again the question would be, what kind of impeller you would like to put and where would you like to put, at what velocity you would like to rotate. So, as to achieve the task of good mixing, ensuring good contact between the various phases, but without spending too much in terms of the cost of the impellerwhich is an initial investment and also the cost of maintenance in the sense of what is the power, which is required to keep these impellers rotating at the given speed. So, if you want to look at overall costfrom a designers point of view, you would have to know how the fluid is taking place within thisand how the solid particles are moving along and what is the distribution that is achieved andunder what ideal conditions optimal conditions can we get the best mixing with as little power consumption as possible. So, in such a case, againyour basic knowledge of fluid mechanics will not be able to give us much information on this.

One may do experiments, but experiments will be very costly. So, again therethere is a requirement for us to know how fluid flows in situations, which do not fall into one of the ideal categories like fully developed flow in a pipe or flow over ainfinitely long flat plate or flow over a sphere or a cylinder which is submerged in an infinite expanse fluid. So, these are all cases which are very simple, but when we look at the practical situation, we may havemuch more complexity.

We can take another example - a common example - let us say that you have ainternal combustion engine and inside that you know that you are sending fuel and the fueltypicallyliquid fuel, which is sent in the form of fine droplets; these droplets would evaporate and then there would be a combustion; and the combustion would release energy, which is then used to drive this internal combustion engine.

As an engineer you would like to know what is happening inside theengine, sothat you can control not only the delivery of the power not only the amount of power that you are able to extract from this, but also you would like to control and minimize the formation of undesirable products like, pollutants,for example, the nitrous oxides; and if you have some sulfursulfurdioxide andthose kind of pollutants and in order to do that, you would have to look at how myhow the fluid is being injectedin to this and how it is vaporizing and what kind ofdisturbance to the flow field that is obtained from this vaporization and how the vaporized fuel and the oxidant that is rest of the air are mixing together; and that is how they are combusting, what kind of heat is being released andhow the heat is being diffused into the rest of thedomain and what are the conditions in which chemical reactions are taking place for this combustion to take place and what are the conditions that are prevalent in which these stray species like, nitrogenoxides are formed.

So, there you are looking at a combination of fluid flow, together with heat transfer together with chemical reaction andlots of time scales and length scales are involved;and this all this is happening while the piston is moving up and down; so, that is a very complicated situation and one can never hope to get any kind ofsimplest solution thatfor this kind of complexity; but if we have generic specialized tools,which constitute the body of computational fluid dynamics at our disposable, then it is possible to represent this transient turbulent chemical reacting flowsimulation to be done in a computer; and from this we can derive some information which willbe useful for us engineers.

We can control therate of fuel injection at which point we inject the fuel,at which point of time we inject the fuel, and in what form we inject the fuel, and how the rest of therest of the power transmission and heat transmission is taking place it is possible tosimulate all these things in the computer. So, that we can therefore, make modifications totothe processes to the process that are in our control to derive the bestperformance from thefrom the engine, whether it is terms of fuel economy or whether it is in in theinin terms of the delivery of the power, the smoothness of delivery of power or in terms of pollutant formation and soon; and there is also theother more chemical engineering aspect of this internal combustion engine that, if you want to put a catalytic converter so as toabsorb the pollutants that are produced in that; then again you are looking at some further chemical reaction that is taking place inside a reactor through which the gases the exhaust gases flow.

So, you have to make sure that,there is good contact between theexhaust gases and theandthe surfaces at which the chemical reaction would take place; and you would also need to make sure that,there is enoughprovision is there for this to happen optimally without too much of pressure drop, so that the back pressure would not bebuilt up to somuch and without anybypassing of the gas;if there is some part of the exhaust gas, which is not able to get into contact with the solid surfaceand if it is not able toparticipate in the chemical reaction, then that part of the pollutant which is there in this bypassing gas will not be converted. So, that will come out in the eventual exhaust.

So, again there is a possible, there is a need for us to for an engineer to know how fluid is flowing throughthis catalytic converter. So, these are these are typical type of fluid flowsituations that are encountered by an engineer inthe daily profession for which one would like to knowanswers to the flow of the fluid; along with the flow of the heat and chemical reactions that may be taking place; and if we need to answer these questions, we cannot rely on what we havelearnt in in our basic courses or advanced courses inin heat transfer and transport phenomena;we have to be able to go beyond what we can do with analytical techniques; so, this is where computational fluid dynamicssteps in.

Computational fluid dynamics, as the name implies,is ais a subject that deals with computational approach to fluid dynamics; and it deals with a numerical solution of the equations which govern fluid flow and although it is called computational fluid dynamics; it does not dealjust with the equations of the fluid flow, it is also generic enough to be able to solve simultaneously together the equations that govern theenergy transfer and also the equations thatdetermine the chemical reaction rates and how the chemical reaction takes place and how mass transfer takes place; all these things can be tackled together in the same overall format. So, this framework enables us to deal witha very complicated flow situationsin reasonably fasttime, such that, we canget a simulation,for example, in ain a few hours' time orin an overnight computation; and for a given setofconditions, an engineer would be able to simulate and see how the flow is taking place and what kind of temperature distribution there is and what kind of products are formed and where they are formed, sothathe or she can then make changes to the parameters that are under his control to modify the way that these things are happening. So, in that sense computational fluid dynamics or c f d becomesa great tool forfor a designer for an engineer.

It is also a great tool forforananalysis for a post-mortem of a of reactor or an equipment which is not functioning well;because in typical industrial applications,many things may be happening and whata designer has hadin mind at the time of fabricating or designing the equipment may not be actually what an operatorof the equipmentintroduces into the equipment at the time of operation, maybe after five years or ten years changes might have taken place in between; and in such a case, the performance of the equipment may not be up to the standard and you would like to modify it in such a way that you can get better performance.

So, the question is then, what has led to the fall in the performance and what kind of measures we can make without making an overall change in theand equipment,withinwhat is within the means of the operator to control; is it possibleto get better performance from the equipment; is it possible to increase the productivity; if you want to look at these kind of what-if kind of analyses, then again computational fluid dynamicswellwell-built computational fluid dynamics model will be able to answer these kind ofquestions.

So, we can see a role for c f d both in the initial design of industrial equipment and also in the day-to-day operation and also in the case ofwhat-if kind of scenario investigations. So, this becomes this makes c f d a very versatile tool in the hands of engineers and also scientists; one would say that,scientists would be interested in knowing,for example, what may be happening in a very small micro scalereactor; if you take micro reactor, one would like to know, what may be happening in that; and it may be very difficult to do experiments in those conditions; and one may be a scientist, may be interested in the very fine transient of a particularphenomenon or a particular situation and it may be very difficult to get anexperimentalnon-intrusive measurement to the same. So, even fora scientist, c f d would makea very good sense as in as a tool for analysis.

So,in that sense, one would say that,c f dis a very useful tool; and over the pasttwo to three decades, c f d tools have become sospecialized and finally, tuned finely honed that they now have become an indispensable tool in the hands of engineers; for a range of industrial applications, all the way from aerospace to metallurgyincludingchemical, and mechanical engineering, civil engineering, environmentalbiological processes and soon.

So, in this course, we are not going to look more on the applications on how we can make use of the c f d codes, but we are going to look primarilyat what how we can get a solution to a set of equations; we are going to look at what kind of equations we need to solve in order to answer these kind of problems and how we can solve these equations using c f d; we will also look at, obviously, why we cannot solve them usinganalytical methods; then we will see how we can solve this usingc f d methods; what we shouldkeep in mind is that,c f d is not mathematics, it is a specialset of techniques which are ideally tuned for the solution ofspecific type of equations that occur in fluid flow is a subject of interest. So, it is not general mathematics that we are interested in, we are interested in in looking at special techniques which have been developed to solve these equations in a very efficient manner;it is those techniques that we want to do. So, we will see that it is nota simple solutionlike that, we would look at what constitutes a c f d solution.

We have a wholly different philosophy of answering a question that,you havesaya particular flow domain what is the flow field. So, we will we will let us start with a simple example, sothat we can try to differentiate between an analytical solution and a c f d approach. So, in this introductory lecture, we will look ata problem which is solved in the c f d way for a very simple problem;for thisproblem, we probably do not need to use c f d, but we will show how we solve it using c f d. So, as to bring out the difference betweenan analytical solution and the c f d solution and then we will elaborate what more we have to do in order to solve the generalproblem. So, at the end of that,we will havea good idea of what c f d is about and we will be able to give an outline of what we are going to do in the rest of the forty to forty five lectures.

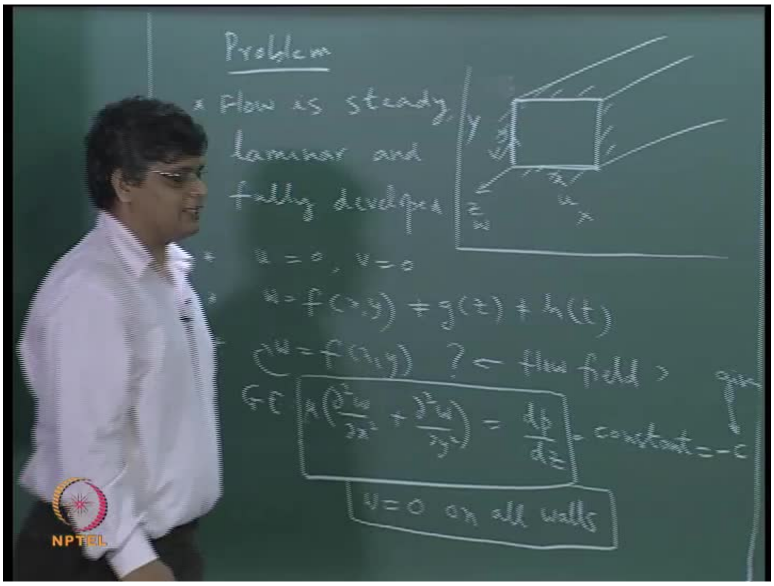
So, let us start with an application of the c f d for a simple problem; the problem that we are looking at isa fully developed steady flow through a duct, which is rectangular in cross section; this is a case for which analytical solutions do exist, but these analytical solutions are obtained in a difficultway,we may have to do conformal mapping of aof thegeometry from a rectangular domain aintoa circular domain; and then we can get a solution to that. So, it is beyond the scope of an ordinary,the general syllabus that is thought inin undergraduate chemical engineering.

So, we will see when we try to use c f d techniques for this;we do not have to do go intosophisticated mathematics, with the simple mathematics that we already know, we can generate a solution, but the method of generating the solution is not somethingthat is enable to handheld calculation; it is amenable only tocomputer-based calculation. So, this is where computational approachcomes into the picture; and it is only with the advent of fast computers with cheap memory thatwe are able to solve a large number of industrial problems using c f d techniques.

If computers had not been there or if computers had been there, but they they were very very expensive, then c f d could not havecould not have spread to lots of industries; it would have still remained in in sophisticatedlabs inelite institutions.

So, it is theit is the bringing together of a robust set of numerical methods for the efficient solution of equations; the easy availability and offast computers with loads of memory storage capability that has really given rise to the development of c f d as an engineering tool.

(Refer Slide Time: 21:37)



So, we will just look at,we will start with looking at a simple case of flow througha rectangular pipe. So, what we are looking at is, a pipe of rectangular cross section. So, this is an infinitely long pipe and we have taken a section here; the flow is fully developed;let us fix somecoordinates;we can put x here y in this direction, and z in this direction. So, this is y and this is z.

We have chosen herea right-handed coordinate system,such that, if you rotate the x axis about the y, the z is in thein thedirection of the right hand threaded screw like this. So, this is thenotation that we have; and along with this x y z notation, we denote the respective velocity components as u v and w. So, the u velocity component is the velocity component in the x direction; v is the velocity component in the y direction; and w is the velocity component in the z direction; and the problem that we have is,the flow is steady laminar and fullydeveloped; and under these conditions, we know that, u is equal to 0 and v is equal to 0 throughout; and we are interested in the flow essentially essentially two-dimensional, in two-dimensional in the sense that,only w is the non-zero component and the w is a function of both x and y and it is not a function of z and it is not a function of time.

So, the flow is steady. So, w is the only non-zero component of the velocity,three velocity components, because its flow issteady and fully developed andlaminar andit is not a function of z. So, at any cross section, theflow field, that is the variation of w with x and y will be the same and its steady and its two-dimensional,in the sense, w is a function of x and y. So, the problem that you want to know is,what is, how does w vary with x and y. So, this is the flow field that we are interested in.

As i mentioned,there are analytical solutions to this, but we are not interested in the analytical solutions; we would like to do this using computational fluid dynamics approach; and this is a simple case, because there is no time dependence and there is only one velocity componentand it is a function of bothonly x and y; and the domain is such that, the flow domain is such that, it fits into a Cartesian coordinate system x yz. So, in that sense,it is a simple problem, and we would like to solve this simple problem for the sake of illustration as to how c f d works.

Now, what do we mean byhow c f d works. So, in c f d wesolve the governing equations using numerical methods. So, first of all we need to find out what is the governing equation or what are the governing equations.

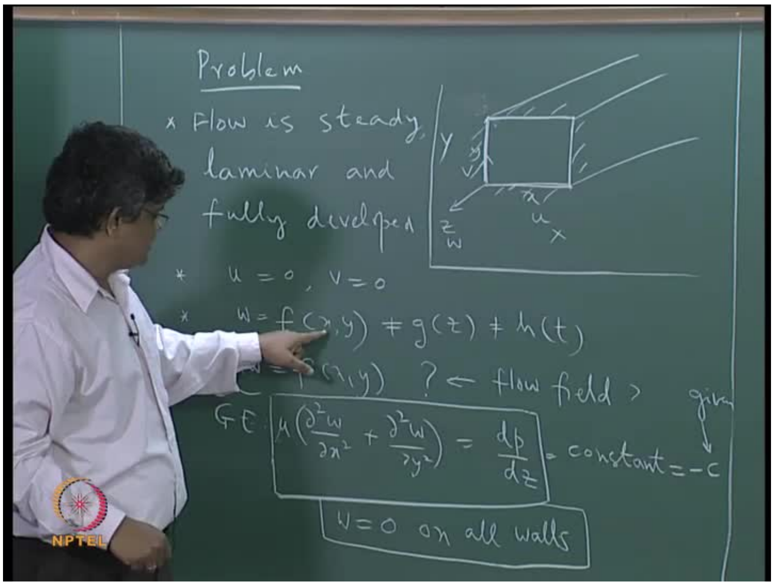
So, in this particular case, we will show later on that the governing equations is given by, we are putting it as g e for governing equationas, dou square w by dou x squared plus dou square w by dou y square times mu, where mu is the dynamic viscosity, is equal to d p by d z, where d p by d z is a is a constant; it is negative in the sense that, as z increases pressure decreases. So, it is a negative constant. So, we can put this as minus c,for example, where c is a positive quantity. So, c is a constant and pressuregradient is constant, because it is a fully developed steady flow inin the rectangular diagram. So, right now we do not have to worry about where this equation has come from;we only say that this is this is a governing equation; and a governing equation is we can see is a partial differential equation. So, this requires boundary conditionssince the flow is steady; there is no time derivative. So, it does not require initial conditions. So, what will be the boundary conditions for this?

We need to look at the variable that we are solving for, the variable that we are looking at is the w component. So, we need to know what is the w. When we talk about boundary conditions, what is the value of w on the boundaries? So, theflow domain of interest is this domain and we know that for fluid flowwhich isbounded by these walls; we know that,the most obvious most applicable boundary conditionis the no slip boundary condition. So, we can say that w is equal to 0 on all walls

So, this is the second part of the specification of problem. So, we now have a mathematical problem which is specified, that is, this equation with a given value of constant; so, this is given here; and with the boundary condition that w is equal to 0 on all the walls of this specifies the mathematical problem.

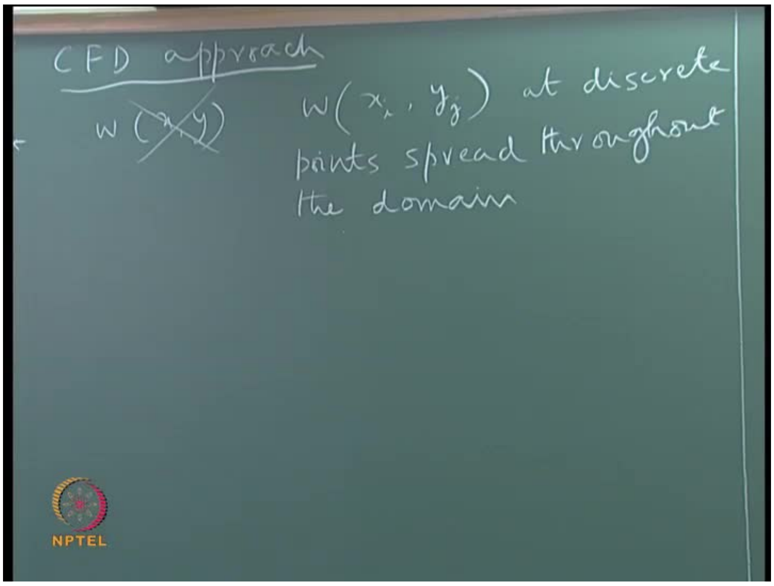
(Refer Slide Time: 29:54)

(Refer Slide Time: 30:26)

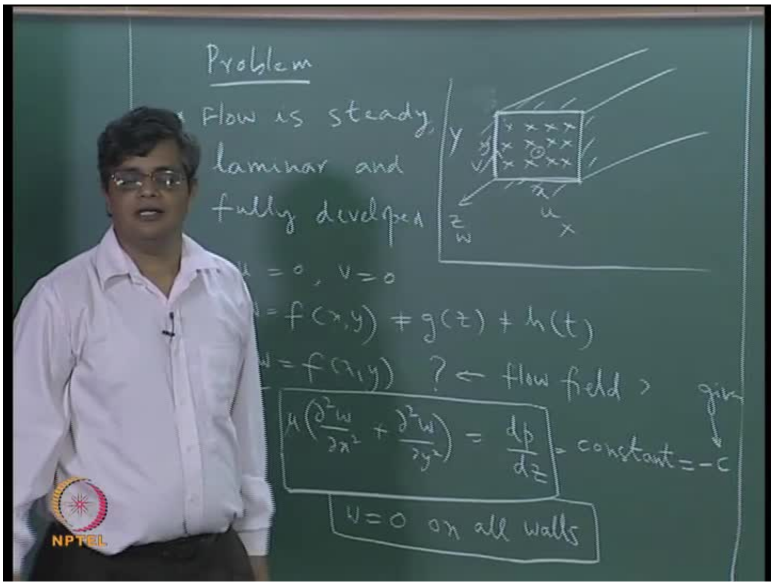


Given this problem we want to know, if this is the casefor a given dynamicviscosity, for a given pressure gradient; and for a given domain,for example, for length, for example, capital x and for height capital y like this, what is then the variation of wat any x and y. So, this is the mathematical problem that we want to solve. So, this mathematical problem is solved in c f d not like this,it is not exactly solved exactly like this. So, at the end of the c f d solution we would not be getting w as a function of x and y. So, in the c f d approach,we do not get wat x y, instead of that we give the value of w at loosely speaking x i and y j,that is where x i and y j are thecoordinates of points, which are spreadwithin the computational domain.

(Refer Slide Time: 30:53)



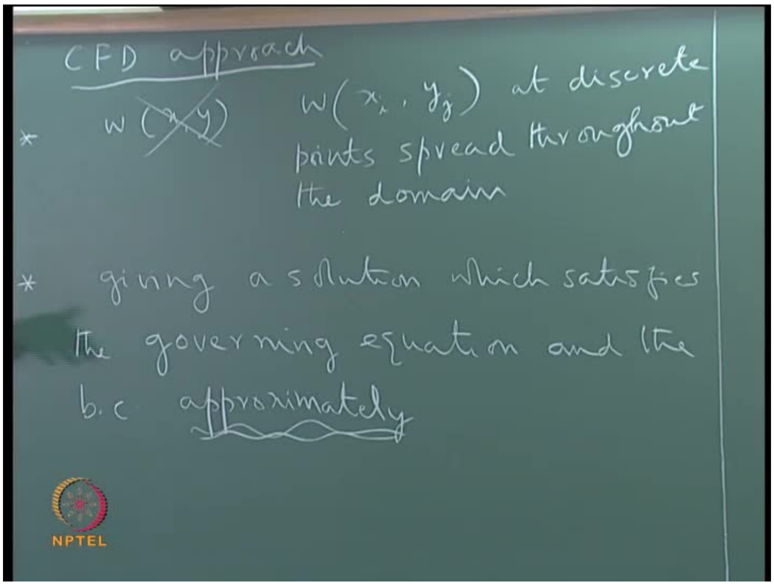
(Refer Slide Time: 31:26)



So, instead of saying, instead of seeking a solution of w as a function of x and y, which is a continuously variable; we are giving a solution of w at discrete points which arespread throughout the domain. So, we give w at discrete points spread throughout the domain; and if you want to have a solution at a point, which does not coincide with any of these points;for example, we havethese things typically we give at several points which are at several points in a fairly orderedlogical sequence like this. So, at these points marked by the crosses we give the solution; and if you are lookingfor a solution at this point, we do not get it from c f d directly, but given that the solution is known at these neighboring points, we can get it by interpolation.

So, we do not derive the value ofa solutionat any x and y; we evaluate the value only at specified points, which are known as the grid points; and so,this we are not getting a continuous solution we are getting a discrete solution, but the points at which we are evaluating are in our points in a way,we decide where to evaluate the points. So, at the same time we do not have total control,for example, we cannot say that, in this overall domain I want at this point and this point and this point; we are forced to have the solution at several points at manymanypoints within the overall domain; typically, it may be a million points within the domain only by putting, only by evaluating the solution at million points,may we have to getcan we getan accurate solution.

(Refer Slide Time: 33:01)



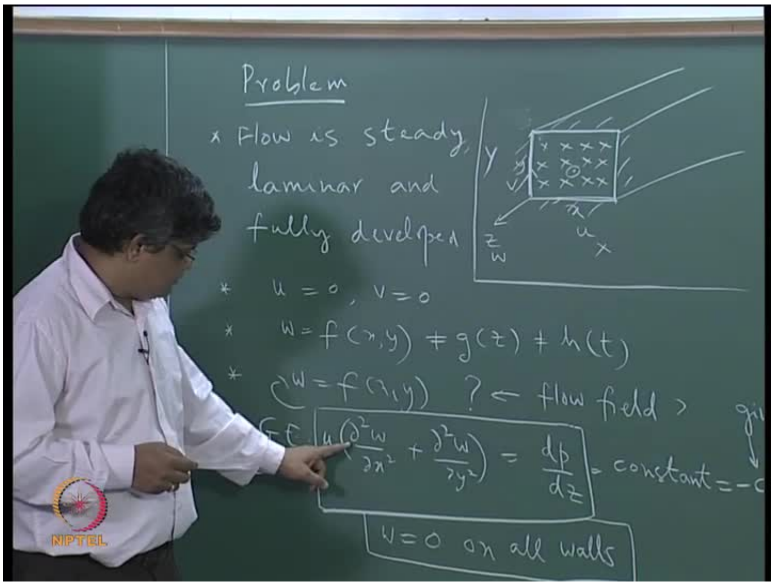
So, the first point we would like to know about a c f d solutionis, the solution is obtained at discrete pre identified points which are spread throughout the domain; and if we want to get a solution at any point in between, we have to do it by interpolationor extrapolation as necessary typically interpolation; and if there are other points of interest,for example, if we want to get the sheer stress of the heat transfer coefficient. So, that information can be derived fromthese values which are obtained as a part of thesolution.

For example, the sheer stress is given by the gradient of velocity. So, since we know the w at several points, we can evaluate the gradient and we can multiply by the velocity and then we can get the sheer stress. So, in that sense we are going from a continuous solution to a discrete solution.

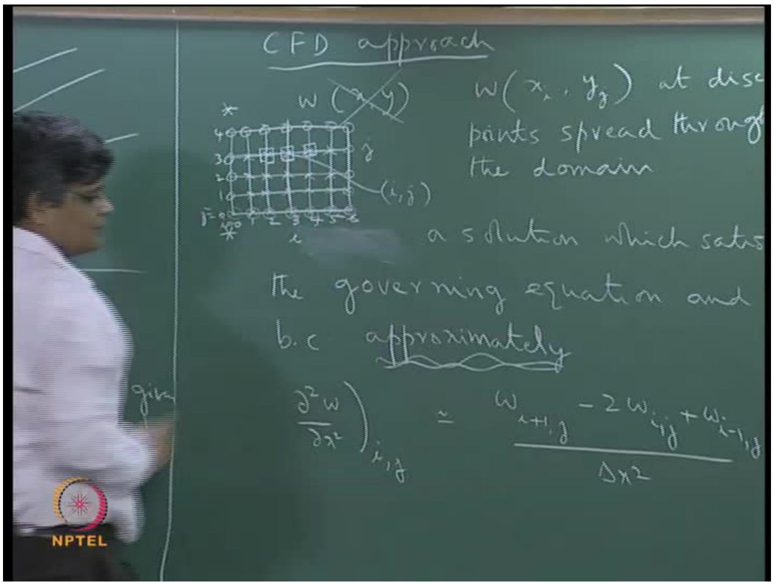
The second point that we want to emphasize about c f dis that, we are not giving a solution which satisfies the governing solution exactly. So, we are giving a solution which satisfies the governing equation and the boundary conditions approximately.

So, we are not getting an exact solution of the governing equation; we are getting only an approximate solution of the governing equation; again here, asthe solution seekers, we havesome control over what would be the solution, we can reduce,for example, the error in the solution that we may be expected, we do not have the exact error, because we do not know the exact solution. So, we can reduce the error the possible error between the c f d solution and the exact solution of the governing equation by choosing a larger number of points or bychanging the way that we approximate the equations. So, we do have a control over howwhat will be the how to reduce the error andthat gives us some satisfaction of getting a solution; but we must keep in mind that, the solution that we are getting from c f d is an approximate solution of governing equation; and it is not guaranteed to bean exact solution may be an exact solution under special cases, but ingeneral, it is an approximate solution. So, how do we generate this approximate solution like this?

(Refer Slide Time: 36:11)



(Refer Slide Time: 36:38)



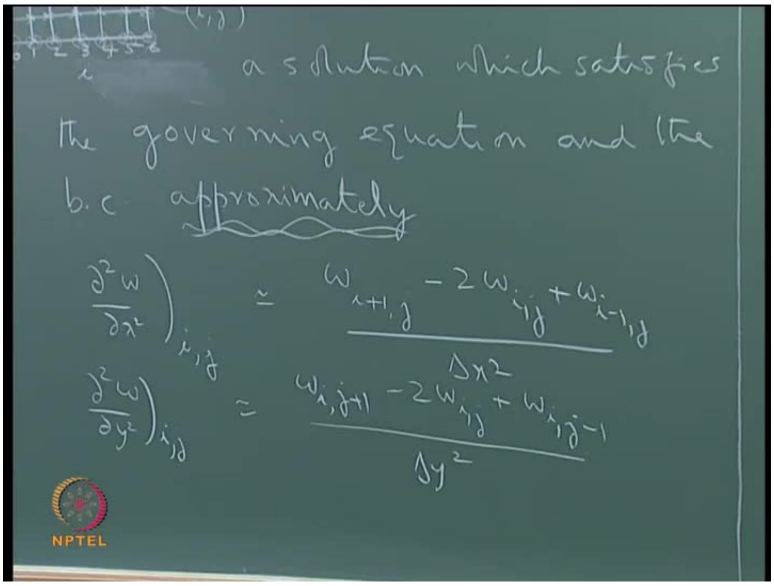
There are several ways of doing it, but for the sake of illustration,what we the way that we try to do is, we thattake the partial derivative here and for each derivativewe substitute an equivalent approximate finite difference approximationfinite difference formula for it;for example, if you have dou square w by dou x square then at a particular point i, j, if this is a point that has to be evaluated; then we can write this roughly as w at i plus 1, j minus 2 w i, j plus w i minus 1, j divided by delta x square.

Now, this requires us to what we mean by this i, j and all this. So, we will go back to this domain here; we choose the points that we want to evaluate in this finitedifference approachat points which are,for example, which are uniformly spread here and which are also intersections of lines of constant x and constant y.

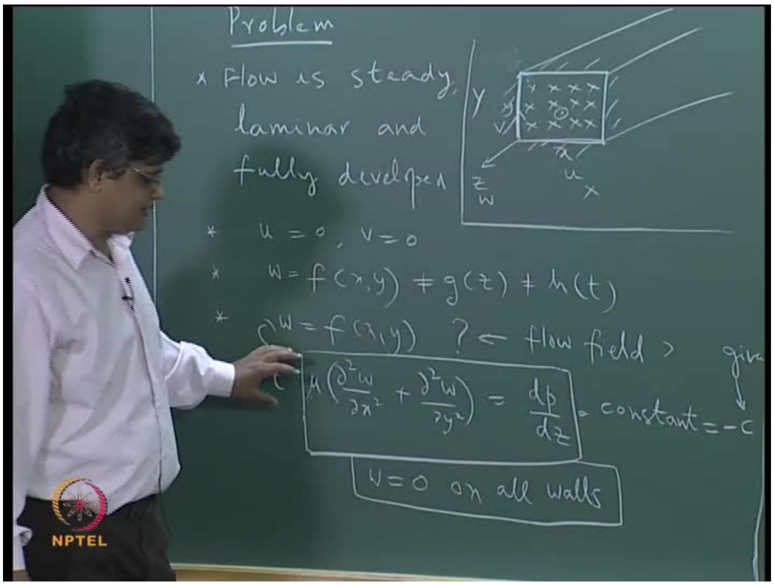
For example, this is line of constant y equal to 0; this is y equal to capital y like that; and this isa line of constant x. So, this point here is anintersection of constant x line and constant y line. So, the i and j here denote the ith constant line in x direction and the j th constant line in the y direction. So, for example, we can put this to be i equal to 0, this is 1 2 3 4 5 and 6. So, we have these lines out of which the zeroth line and the sixth line constitute the left boundary and the right boundary of this; and similarly, we can put j equal to 0 as the line corresponding the bottom boundary and this is the 1 2 3 4. So, we have divided the domain intoone two three four five six divisions in the x directions and four divisions in the y direction. And so, as they are at the intersection of each of these lines, there are points and at these points we want to evaluate the w; andand we see that some of these points lie on the boundary; and at that point, we already know that w is equal to 0.

So, this is known as a dirichlet problem, where the value of the variable that we are evaluating is specified in the boundaries; so, in that sense these are the values, these are the locations at which we do not need to evaluate w, becauseit is already known at all these points; we do not need to evaluate, we only need to evaluate the values in the interior. So, when we say, we can now takeith x equal to constant line and a jth y equal to constant line. So, this is an ithx equal to constant line; the intersection of this here, this point is the i, j point. So, the point is denoted by theindex i denoting the ith coordinate constant coordinate line in the x direction and j indicating the constant coordinate line in thej direction. So, under these conditions assuming uniform spacing in the x direction and uniform spacing in the y direction, we can represent, we can writean approximate formula for the second derivative in thex direction like this, where this value is being evaluated ati j and this is i plus 1 j. So, it is now we are looking at this particular point. So, this is w i plus 1 j means this point here, w i minus 1 is this and w i j is this. So, we have, we are approximately writing,we have a formula for writing this and this is a formula which is one of the many formulas and we will bederiving how to do this approximation later on we will be deriving this inlater parts of the course.

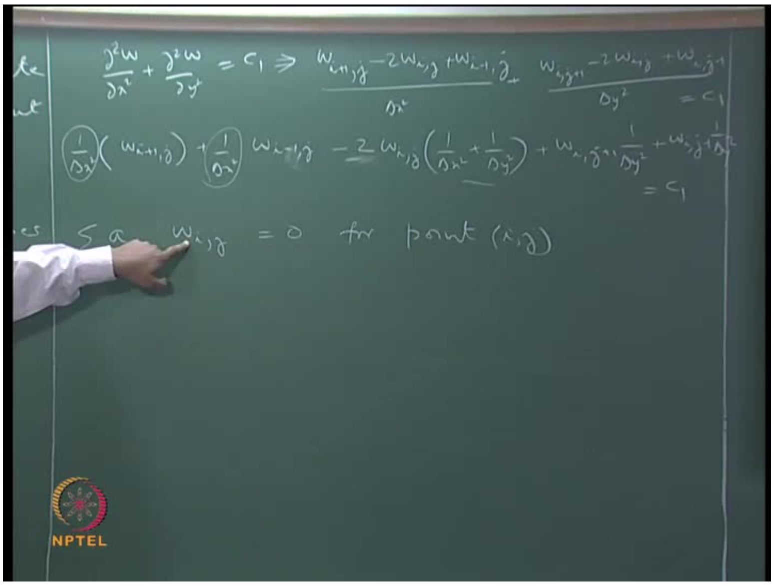
(Refer Slide Time: 41:46)



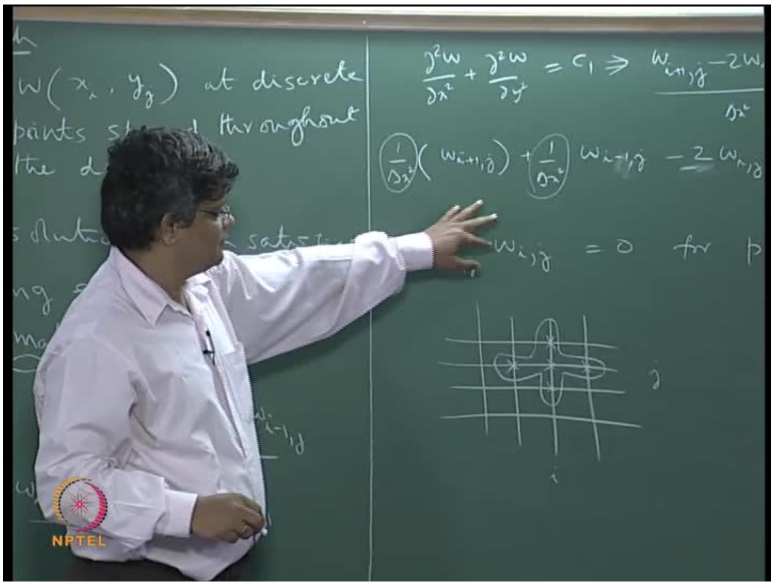
(Refer Slide Time: 42:21)



(Refer Slide Time: 43:03)

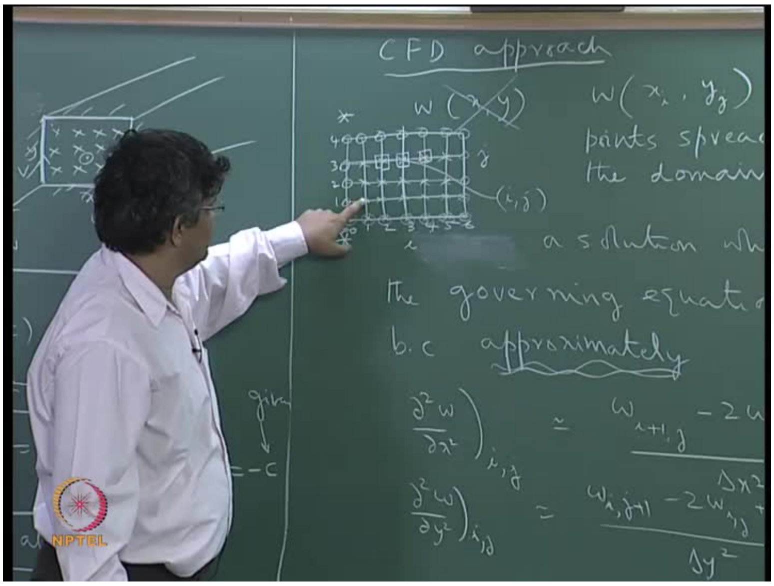


(Refer Slide Time: 46:30)



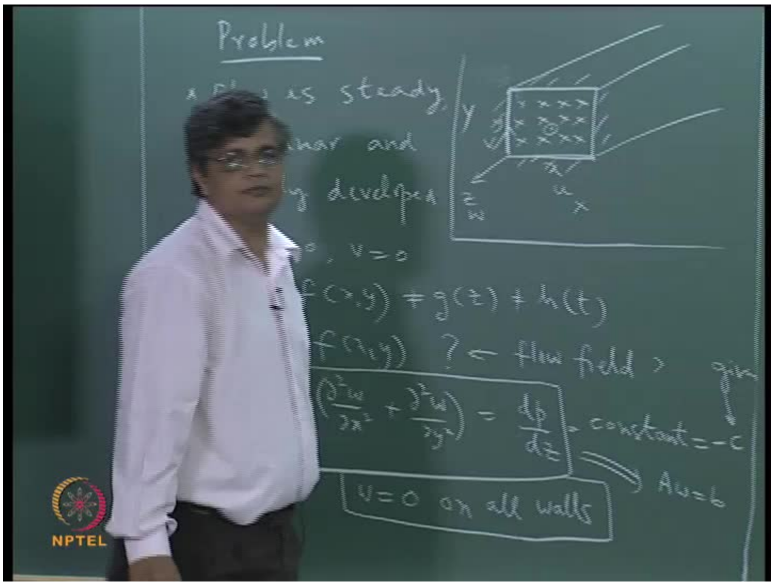
Similarly, we can write another approximation for this,dou square w by at the same point i, j can be written as w i j plus 1 minus 2 w i, j plus w i j minus 1divided by delta w square. So, this again is an approximate formula for the dou square w by dou y square at i, j. So, now, we can take thisequation here; replace this derivative by this formula; and replace this derivative by this formula; if we do that, what we will get is an algebraic equation; the resulting equation will not have any partial derivatives, it will only have thesevariables w at several points within the domain of interest. So, let us do that; we will take for the sake of simplicity mu here andput this as c by muas some constantk c 1. So, we can say that, dou square w by dou x square by dou square w by dou y square equal to c 1, where c 1 is 1 by mu d p by d z, we can now be written as, w i plus 1, j minus 2 w i, j plus w i minus 1, j by delta x square plus w i j plus 1 minus 2 w i j plus w i j minus 1by delta y square equal to c 1; here delta x is the spacing in the x directionand delta y is the spacing in the y direction which is known; c 1 is a constant, again which is known; and again we can rearrange this and write it as 1 by delta x square wi plus 1, j plus 1 by delta x square wi minus 1, j minus 2 by 2 w i, j1 by delta x square by 1 by delta y square plus w i j plus 1 by deltay square plus w i j minus 1 1 by delta 1 square equal to c 1; and what we see here is, these are the variables and each of them has a coefficient 1 by delta x square like this these things the coefficients are something that we can do. So, we can rewrite this expression as a i jis equal to 0,where this is this is an algebraic equation; and this algebraic equation hereis an equation forpoint i j. So, in this equation, there arethe coefficients here 1 by delta x square delta y squareand all those things; they depend on theparticularvalues of the variables that come in this particular approximation. So, if you were to look atthis case, we will just takea generic points like this, these are lines of constant x, these are lines of constant y here.

(Refer Slide Time: 47:19)

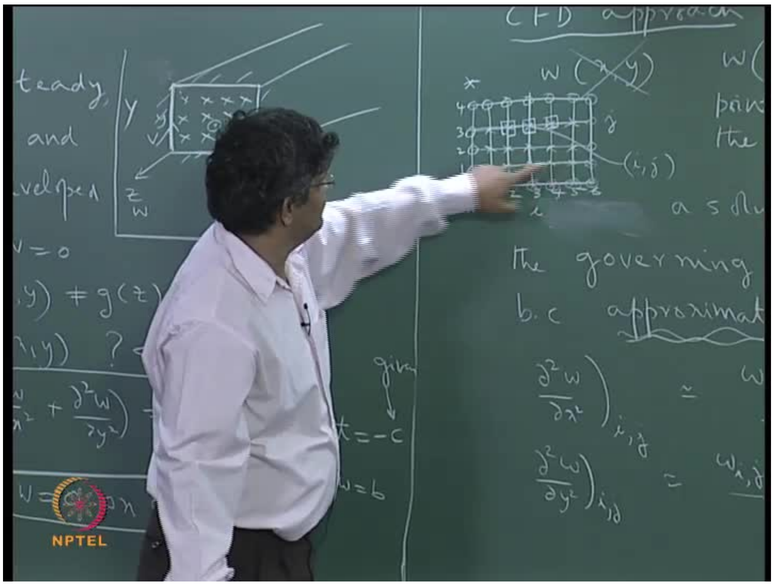


If you are looking at i here, and j here. So, this is a point that we are looking at. So, we can see that, this hasthis value is appearing and the neighboring value to the right and the neighboring value to the left and the neighboring value to the top and the bottom. So, these five are thevariables unknowns which are appearing in the equation for the point i, j; and similarly, we can writederive a similar equation for all the points at which we want to know, we want to find the value of variables; so, that is for all these points; at each of these points we writethe same finite difference approximation for the derivative and each of those approximations will give us an algebraic equation like this.

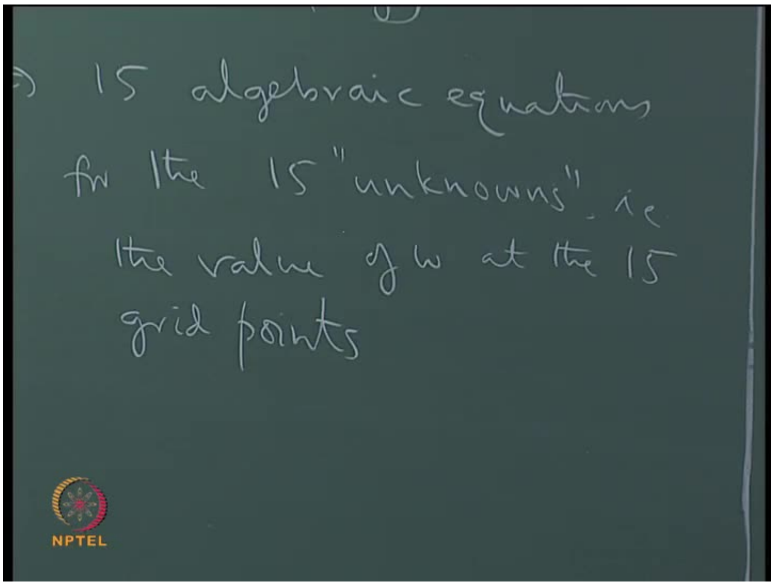
(Refer Slide Time: 48:01)



(Refer Slide Time: 48:15)



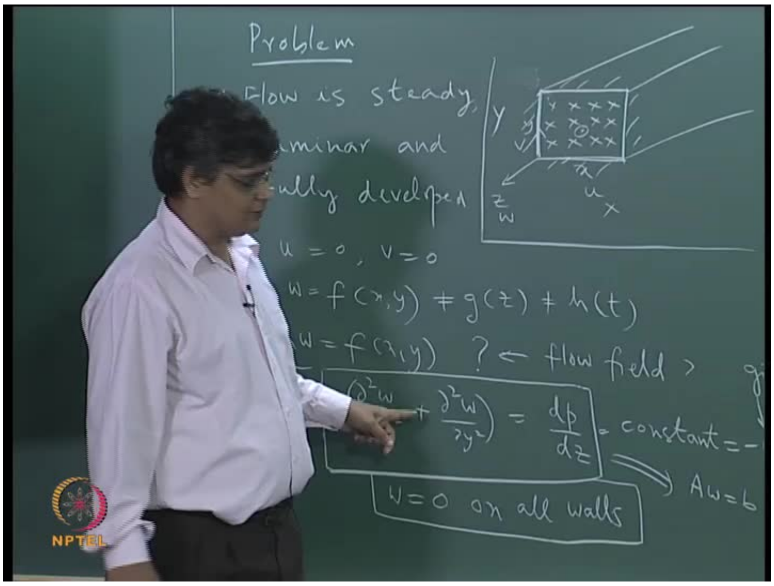
(Refer Slide Time: 48:28)



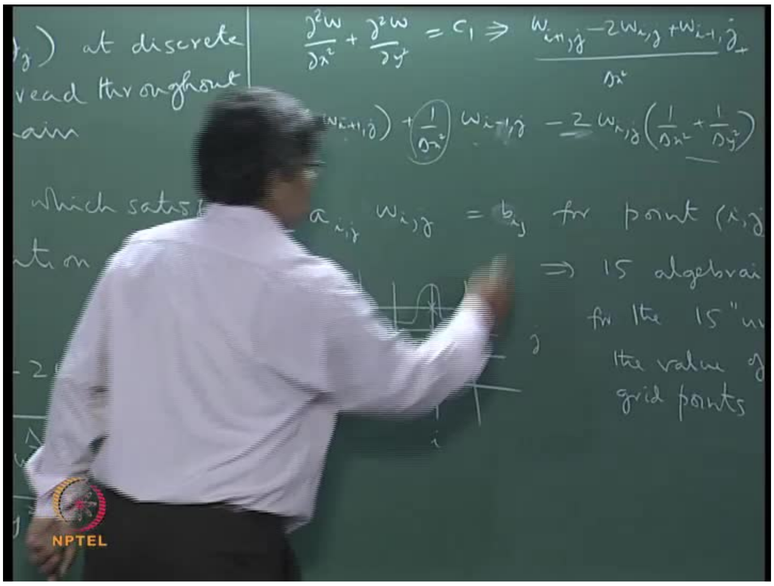
So, if we have here we have fifteen points'algebraic equations for the fifteen variables w at this point, this point,this point, like this. So, at the end of that, this partial differential equationis converted in to an equation like a w equal to b, where a is the coefficient matrix, and w is the set of unknowns, that is w 1 1 2 1 3 1 4 1 5 1 and then1 2 2 2 3 2 4 2 5 2 like this. So, we they will have a set of fifteen equations, fifteen algebraic equations for the fifteen unknowns; and the unknowns are the value of w at the fifteen grid points.

In this simple case, the fifteen equations that we are getting are linear equations; linear equations in the sense, equations in which these are the variables which have the coefficients of which are constants, delta x is constant, delta y is a constant. So, all these things are constant.

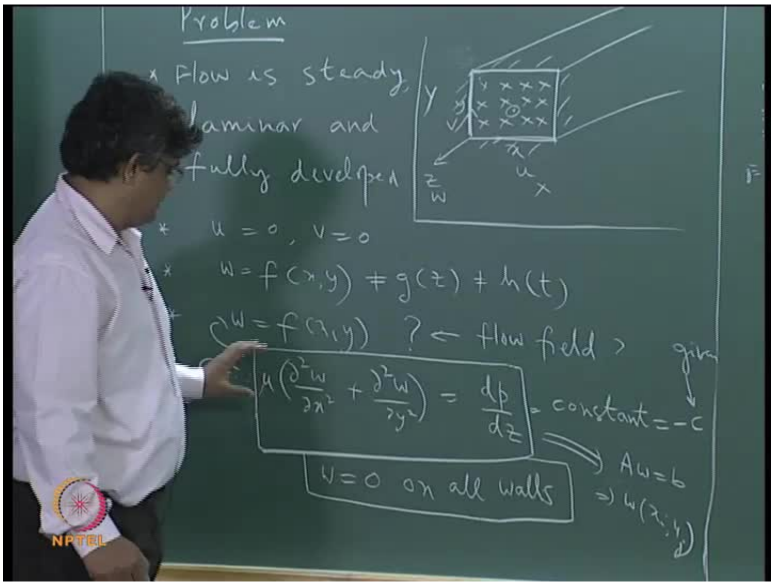
(Refer Slide Time: 49:31)



(Refer Slide Time: 49:58)



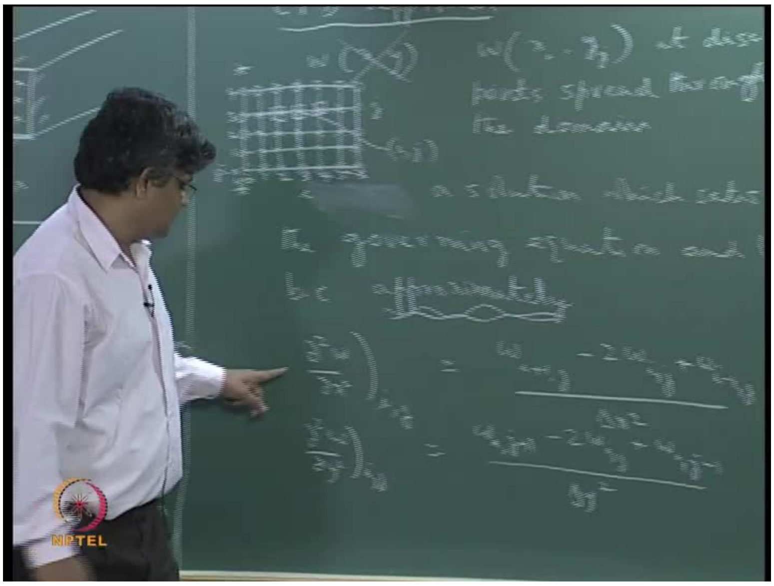
(Refer Slide Time: 50:08)



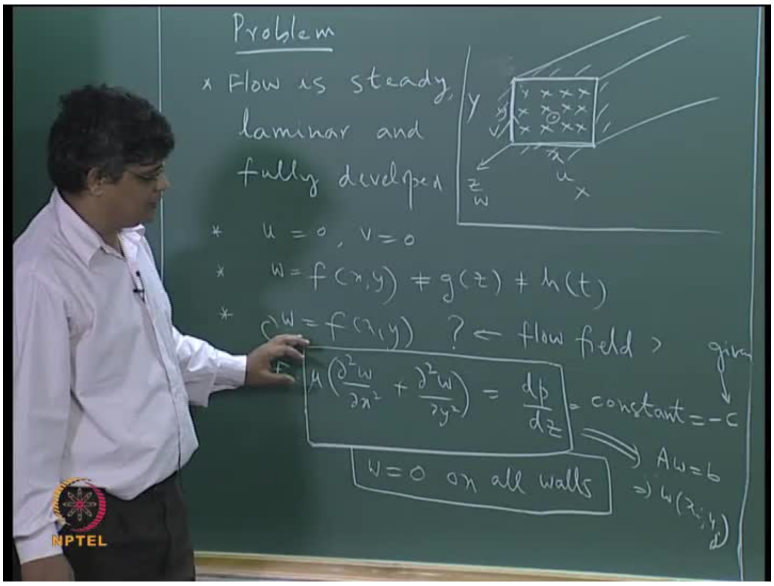
So, what we have ended for this particular problem; this partial differential equation is computed to a matrix witha matrix of fifteen variables with constant coefficients; and as part of this substitution which is known asa discretization step, we not only have the value of the coefficients a i j, but we also have this is not zero, here this is let us say b i j. So, we also have the value of the b for each equation; and there are several methods for the solution of these matrix equations; and there are, for example, direct Gaussian elimination is one equation, which is fairly widely practiced and then we have iterative methods like Jacobi method and gauss seidel method and soon. So, there are several techniques for the solution of a w equal to b when we do this inversion. So, from this we can get w that is we can get w at x i y j. So, this is how we are getting a solution in the c f d approach.

We are not solving equations directly, in the sense that,we are not, for this governing equation and for this boundary conditionwe are not getting w at any x y, we are getting only at the selected grid points and not only that we are not getting thevalue of this w at x l y j, but by solving the exact equation.

(Refer Slide Time: 51:26)



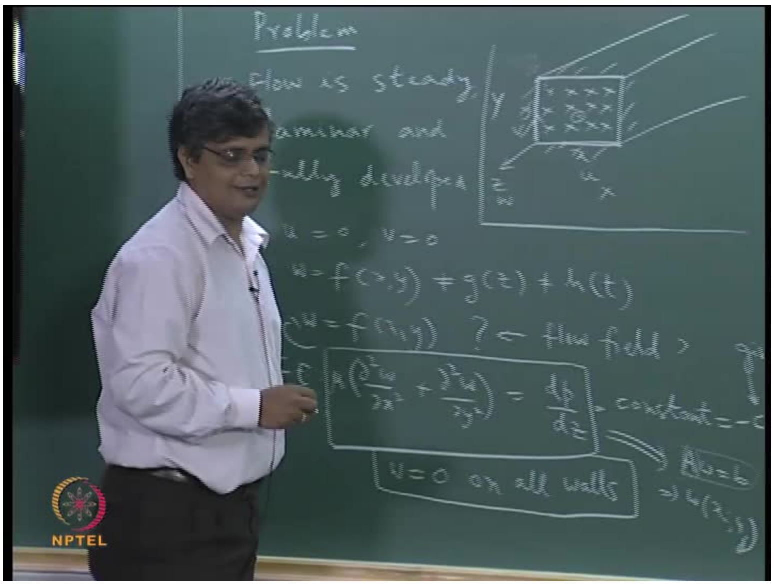
(Refer Slide Time: 51:33)



We are solving only an equation which is an approximation of this governing equation, because we have made an approximate formula for thetwo derivatives. So, in that sense, in the c f d approach we convert the governing equation in to an approximate set ofalgebraic equation; and then we solve it tomachine accuracy or to some degree of accuracy usingnumerical methods and that will finallygive us the solution at the preselectedgrid points, which are spread throughout the domain. So, this is we canhow the c f d approach works; and we can see that in the process of generating this solution, we have not really useda lot of mathematics that we are notunaware of. So, these approximations finite difference approximations are fairly well known; and the conversion of substitution of this and conversion of this in tothe matrix equation is just a question of good book keeping it is not very difficult, once we evaluate these formulas and the solution of the matrix equation is also not very difficult.

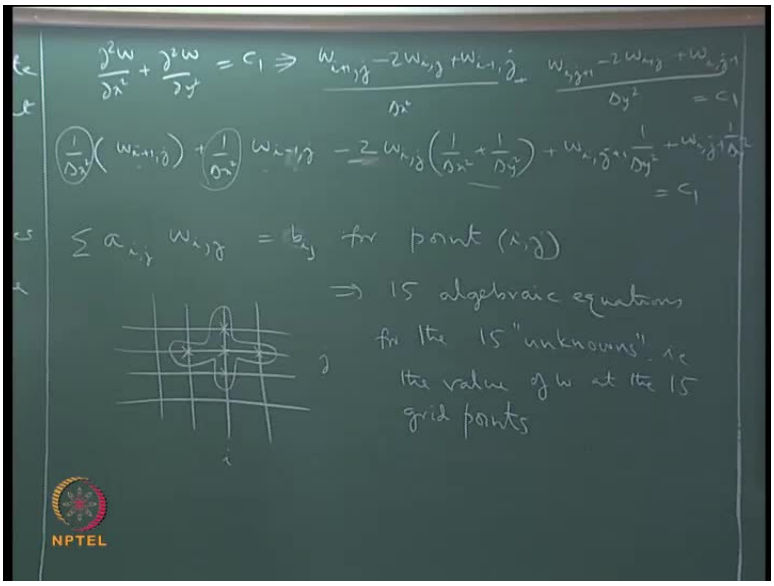
So, in that sense, we are not on unfamiliar territory for this simple case, but when we look at the general case, when we look at the real three dimensional turbulent reacting flow case, then many of these solutions becomemuch more complicated andthat is where c f d comes intopicture; because for a simple problem, we are looking at fifteen equations, but if you want to get good accuracy thenwe need to put lots of points; we would ultimately like to get a solution of w x and y which is very accurate; so, that means, the approximation of this p d in to the correspondingdifference approximation is this has to be very accurate; and that is possible if you have large number of points, the more the number of points the better will be the accuracy.

(Refer Slide Time: 53:41)

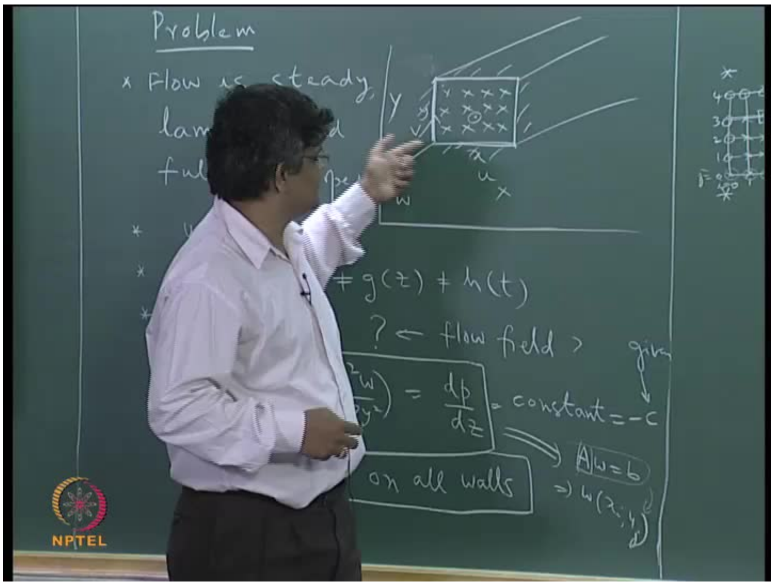


So, we should have large number of points; and what that meansis that, the size of this matrix equation, the size of thiscoefficient matrix a becomes very large; and instead of fifteen equation we may have fifteen hundred equations or fifteen thousand equations.

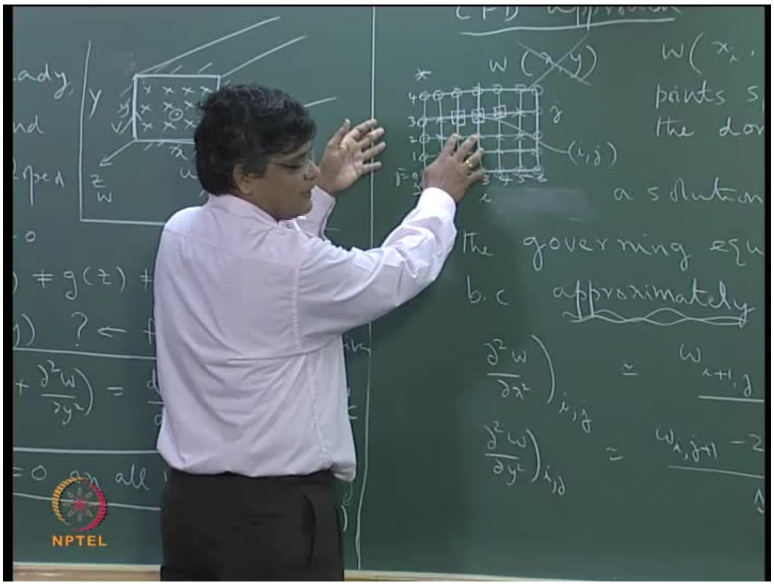
(Refer Slide Time: 55:10)



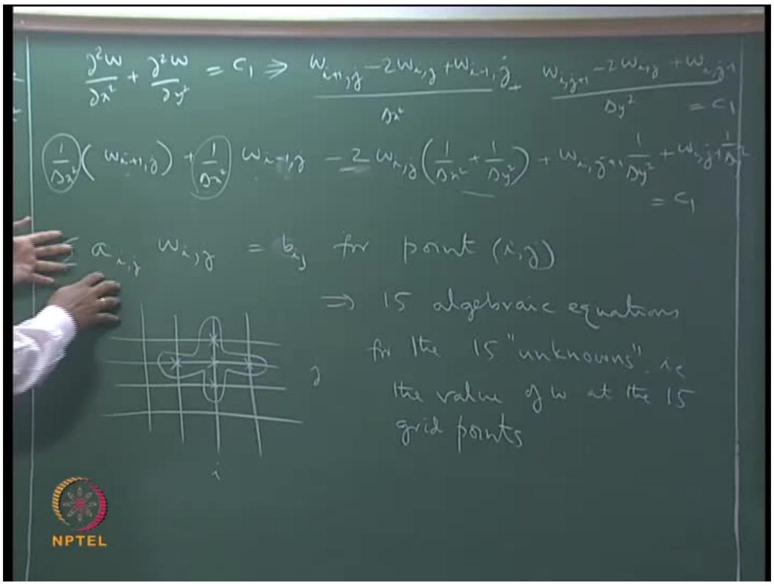
(Refer Slide Time: 55:27)



(Refer Slide Time: 55:48)



(Refer Slide Time: 55:57)



So, as the matrix size increases, the solution of this herebecomes very problematic, it becomes very time consuming. So, we need to have very efficient solutionsmethods for the solution of these equations. So, that is one difficulty; then you have a more general problem, you normally do not have a single equation, you have three equations,for example, for a general two-dimension flow or four equations for a general isothermal non-reaction flow a lamina flow; so, that means, you do not have a single reaction, but you you have four variables and we have to solve that thing four times and not only thatwe will see that the equations are not in such a way that we can solve them separately; if we want to solve the four equations, which govern, which are called the navier stokes equation; you cannot solve them individually, you have to solve them together and that makes it a problem for the solution. So, not only that, in the general case, this sort of approximation does not give us linear equation; for the simple case of this type of posa equation we get a linear algebraic equation; and in the general case, we have non-linear algebraic equations for which the solution of which is even more difficult; and when you look at a general domain and not something that fits in to this x y z coordinates like this, then putting identifying the points at which we want to get the solution and spreading them throughout the domain that itself becomes a project. So, every aspect of this solution, that is, from the equations and finding the corresponding approximations and spreading the points throughout the domain converting thisequation intoan algebraic equation and then the solution, every partof this becomes much more difficult as we go to the general case; and this difficulty iswhat has drivena lot of developmental activity towards finding specialized algorithms, which are needed for each of these steps; and that is what constitutes the body of c f d; and that is what we should be aware of in order to get a c f d solution for the general case; when we do that, essentially, the world is atour feet; and we can use these techniques for the general case of fluid flow in any type of situation.

So, in the next class we will illustrate one method,which is,which can be used for the solution of these equations; we will work out an actual problem, sothat we canget a real feel for this; and thenwe will take it further to discussthe case of genericgeneric fluid flow solution.

Thank you.