Turbomachinery Aerodynamics

Prof. Bhaskar Roy
Prof. A M Pradeep
Department of Aerospace Engineering
Indian Institute of Technology, Bombay

Lecture No. # 40
CFD for Turbomachinery:
2D and 3D Blade Generation and Analysis using CFD

We have come to the last lecture of this lecture series on Turbo machinery Aerodynamics. We have gone through all the components of machines that are called Turbo machineries, and the aerodynamic principles of those components, and in the last few lectures, we have been taking about application of Computational Flow Dynamics-CFD in this exciting field of Turbo machinery Aerodynamics.

So, in the last lecture today, I will be again talking about CFD, and its application to Turbo machinery. And will end up with some idea about how to apply CFD in turbo machinery blade design. We have talked about blade design earlier of the compressors of the turbines, and will have a quick look at how CFD is used in Turbo machinery blade design. You have already had a couple of lectures in which CFD has been introduced to you, these lectures which were done earlier, and today's lecturer essentially, introduction to CFD in the field of Turbo machinery.

If you do wish to actually, become an expert in the field of CFD of Turbo machinery. It is necessary that you go through a full course in CFD - a separate course. And then possibly learn various techniques that are required for application of CFD to Turbo machinery. As you have already seen in the last couple of lectures, application of CFD in Turbo machinery is some more different from application of CFD in many other areas. So, Turbo machinery has its own aerodynamic issues, problems, challenges and some of the problems are not yet properly resolved. So, we are looking at issues that probably have some futuristic implications. In the sense, that CFD is still a very challenging and open field of research in the field of Turbo machinery.

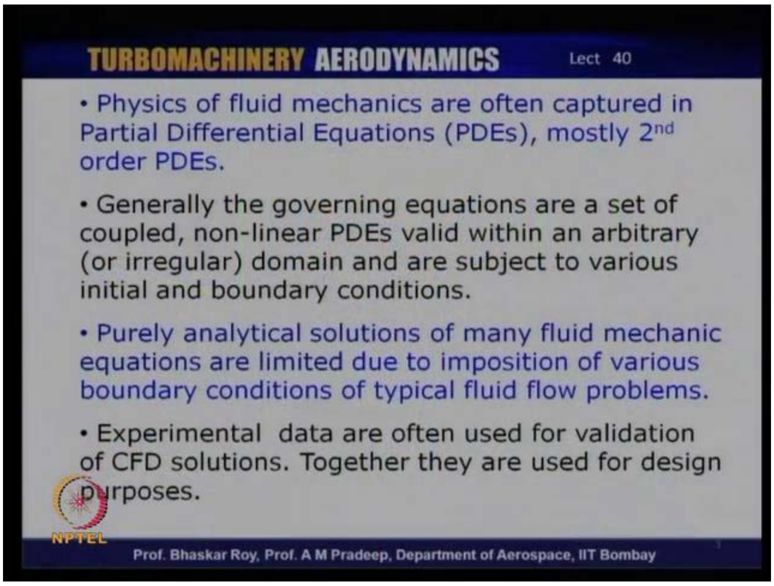
In today's lecture we will take a look at some of the fundamental issues of CFD. A bit of through back in to the fundamentals, and then we will move into how to apply CFD, a little bit of which you have done in the last couple of lecturers, and then apply try to apply that to what we have done earlier in the field of blade design. So, let us first start of with the fundamental issues of CFD, which you probably would learn much more in much greater detail, if you get in to a full-fledged course of CFD. Let us, take the fundamentals today first.

The fundamentals of CFD are have been around for quite some time, almost twenty five 30 years actually, a little more if you would like to think of right from the beginning. And some of these are based on fundamental mathematical principles. Now, these mathematical principles have been around much longer, and for a long time people were trying to figure out, how to find solutions to those mathematical equations, which try to capture some of the aerodynamic issues or the physics of the aerodynamics under certain given circumstances.

The problem was that many of these mathematical equations, which are let us see in the form of ordinary differential equations-OD is. The analytical solutions to those often are simplistic; and not necessarily very useful especially in the field of Turbo machinery; a little more comprehensive one, which are captured in Partial Differential Equations or PDE. These have some problems, in the sense that straight forward analytical solution is often not possible. And hence, one resorts to various numerical methods. Now, this numerical method is what finally constitutes computational fluid dynamics. In this numerical method essentially, the Partial Differential Equations are converted to algebraic form, and then simultaneous solution of these algebraic equations quite often number of equations, for example, as you know in flow dynamics, you have the energy equation, you have the moment equation and the continuity equation. So, you try to solve number of these which are applicable to your field of application, of the physics, of aerodynamics, as you are doing now, and when you apply these you try to resort to numerical solution which is essentially as I mentioned, involve converting you have PDE to algebraic form, and trying to find solution to this algebraic equations.

So, we will take a look at some of these fundamental issues very quickly, very briefly, because it is a very big field, indeed very exciting field, and we can only take a very quick look into some of the fundamental principles that covered CFD. Let us take a look at some of these fundamentals.

(Refer Slide Time: 06:25)

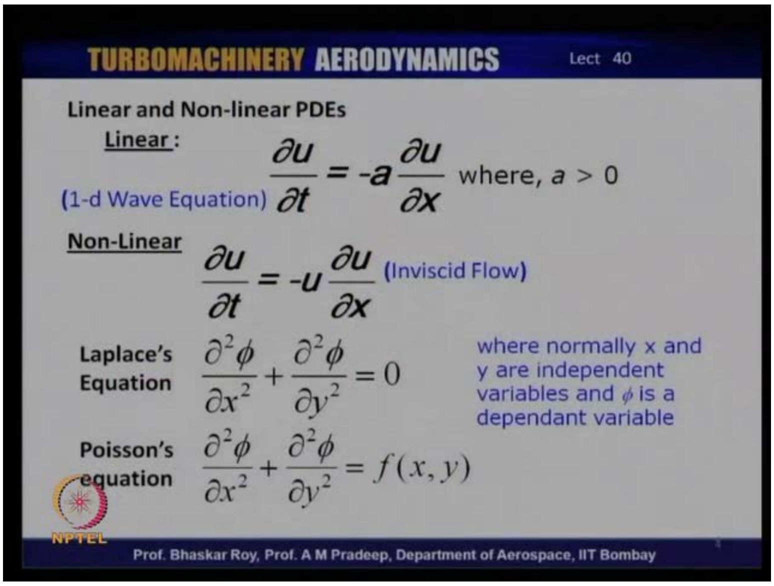


So, as I mention the physics of fluid mechanics are often captured in Partial Differential Equations, and most of the useful ones are in second order Partial Differential Equations. Now, the generally the governing equations, as I was just mentioning few minutes back, a set of coupled non-linear Partial Differential Equations valid in an arbitrary or irregular. Let us say, undefined domain, and subject to various initial and boundary conditions. We will be talking about these initial boundary conditions little, in a little while, so some of these definitions need to be understood in proper context. Now, as I mentioned the purely analytical solutions of many fluids mechanic equations are limited due to imposition of various boundary conditions of typical fluid flow problems.

So, very simplistic solutions are available analytically, and they give you what can be called very simple handy solutions, and beyond a certain level those handy solutions have very limited utility value. On the other hand, there is reason to believe that experimental data are often used, because it is there is reason to believe that CFD is not here in a position to capture everything that is happening inside, especially in Turbo machinery. So, quite often at the end of the day, you probably need to have some experimental data also, and together they form that so called analytical drop back drop, which the designer could use for finally designing a product in our case Turbo machinery.

So, experimental data definitely is required. It is simply called validation. You need to have validation of CFD three experimental work. And once they are validated or they are matched, then you can say that you have a data that is reliable, and the data can be used for design purposes. So, CFD alone often does not quite produce sufficient information, it needs to be back tap, because it is known that under certain circumstances the solutions or the results or the output that is available from CFD solutions are not necessarily the correct solutions. So, some of these things would need to be understood depending on the circumstances, and then the CFD data would have to be as I mentioned validated and coupled or complimented with experimental data, and then you have a certain set of data that are said to be useful for design or final analysis purposes. So, let us take a look at the fundamental issues of Computational Flow Dynamics.

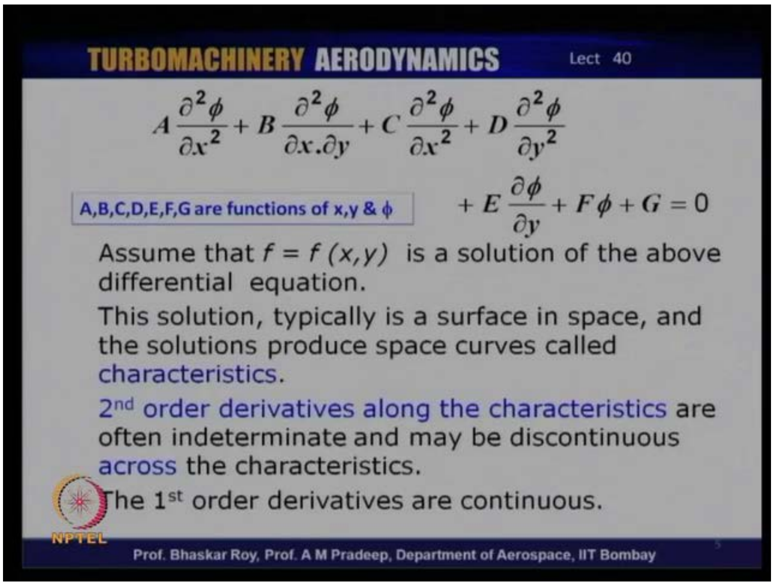
(Refer Slide Time: 09:45)



Now, as I mentioned the they are often the flow dynamics is often captured in Partial Differential Equations. Now, they are two kinds really; the Linear and the Non-linear. Once the Linear one that is shown in over here is a 1-d - one-dimensional Wave Equation, very well known. And that is del u del t equal to minus a del u del x, where a is normally greater than 0.It is a non 0 quantity. On the other hand, a Non-Linear Equation of the would be del u del t equal to minus u del u del x; this is often the barges equation, and is a often apply to inviscid flow that means there is no viscosity in that flow, what one sometimes is also called ideal flow.

The other form of Non-Linear Equation is other well known equations - the Laplace's Equation del square phi del x square plus del square phi del y square is equal to 0, where x and y are independent variables, and phi is a dependent variable of in the potential. And then of course, the Poisson's Equations which is del square phi del x square plus del square phi del y square is equal to function of x and y. So, these two are very well known equations quite often used to capture somewhat simpler versions of fluid mechanic a problems, and often give handy quick solutions to simple fluid mechanic issues.

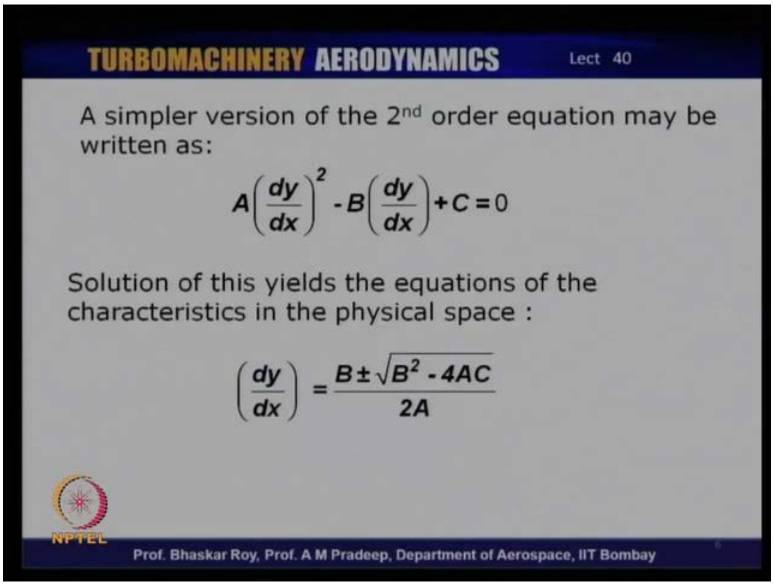
(Refer Slide Time: 11:28)



Now, if we move on to a little more generalized form of the Non-Linear Equation; and if we just write down that the whole equation in the form of A into del square phi del x square plus B into del square phi del x del y plus C into del square phi del x square plus D into del square phi del y square plus E into del phi del y plus F into phi plus G equal to 0. In this generalized form of Non-Linear Equation A,B,C,D,E,F,G which are written down here are essentially functions of X and Y, which are independent variables, and phi which was a dependent variable.

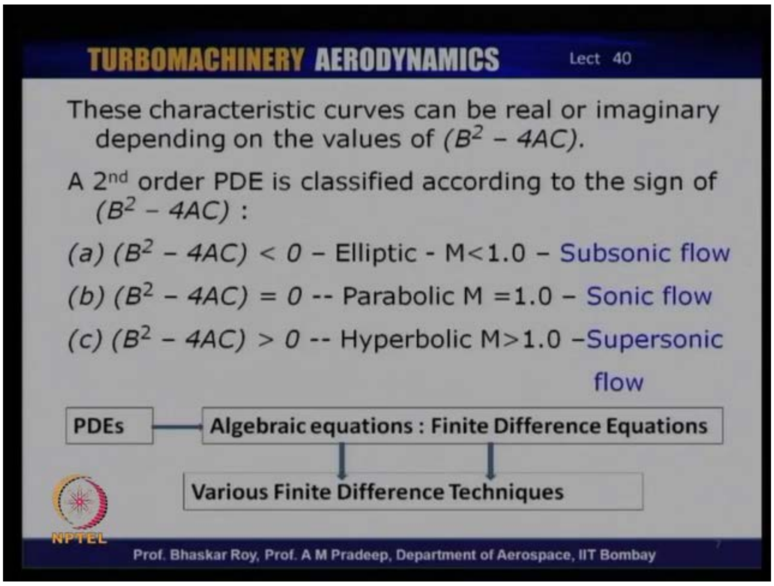
Now, if we assume that f equal to f X,Y that is function of X Y is a solution of the above differential equation. This solution would typically be is would be a surface in a space, and the solutions produce space curves which are called characteristics. So, quite often you get characteristics as a solutions, which essentially come out of above a Non-Linear Equation. Now, the second order derivatives along the characteristics are often indeterminate. And they may even be discontinuous across the characteristics. On the other hand, the first order derivatives are continuous. So, these are some of the simple solutions that you can get out of Non-Linear Equation.

(Refer Slide Time: 13:11)



Now, a simpler version of the second order equation may be written down as A into dy square by dy dx square minus B into dy dx plus C equal to 0. Now, solution of this second order equations yields the equations of the characteristics, in the physical space, the characteristics we are talking about in the last slide, and that would be dy dx equal to B plus minus root over B square minus 4AC by twice A. Now, this is of course the standard solution of a second order equation, many of you might recognize this standard solution of an algebraic equation of second order, so it is something of the same order.

(Refer Slide Time: 14:06)

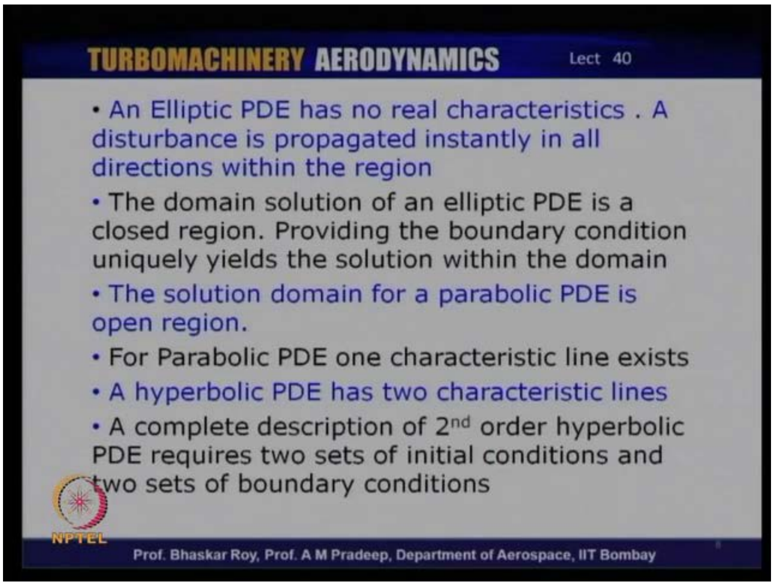


Now, this gives us the characteristics. Now, these characteristic curves can be real or imaginary. As we saw in the solution depending on the values of B square minus 4AC. A second order Partial Differential Equation is normally classified according to the sign of B square minus 4AC. Now, these are conventions. So, the classification is that if B square minus 4AC is negative that is less than 0. This is an Elliptic form of Partial Differential Equation which is normally applied for Subsonic flow that is Mach number less than 1. So, when the Mach number is clearly less than 1, and which in Aerodynamics, we simply say that there are no shocks anywhere around. The Elliptic form of the equation is often applied for finding solution to those Aerodynamic problems or Aerodynamic issues.

On the other hand, when B square minus 4AC is equal to 0, we call the Partial Differential form of the equation as Parabolic form and it is applied for Sonic flow, where Mach number is exactly equal to 1. And then, when B square minus 4AC is greater than 0 that is its positive, the Partial Differential Equation is referred to as Hyperbolic equationm, and it is applied to Supersonic flow, where Mach number is clearly more than 1. So, you have different forms of Partial Differential Equations to be used for Subsonic flow, Sonic flow, and then Supersonic flow. It is stands to reason that when you have both kinds of flow, Subsonic and Supersonic existing in a particular flow domain of interest, you would probably need to apply both the forms in these two different zones in a judicious manner to get you a correct solution.

Now, this Partial Differential Equations, as I was stating a while back are to be converted to algebraic equations, which then of course, are converted to Finite Difference Equations and then Various Finite Difference Techniques, the forward difference, the backward difference, ectera are often use to find solutions to the Algebraic equations. So, what you are finally solving are a set of up algebraic equations, which have been evolved from the set of Partial Differential Equations. So, that is the procedure that computational flow dynamic people are often adopt.

(Refer Slide Time: 16:52)

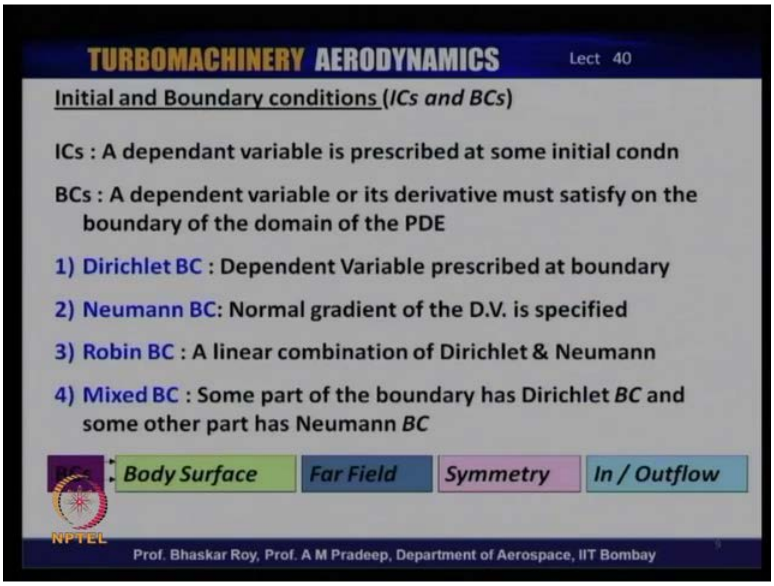


So, the Elliptic Partial Differential Equation has no real characteristics. We were talking about the characteristics. A disturbance if it is propagated in the domain, it it propagates instantly in all directions. Otherwise, if you have characteristics, it disturbance propagate along the characteristics. A domain solution of an Elliptic PDE is a closed region. And normally, you get a closed solution. And the providing the boundary conditions uniquely yields the solution within the domain. So, it it is easier to get the solutions in the Elliptic domain in a very well understood form.

The solution domain for a parabolic PDE is open region, parabolic we saw was applicable to Sonic conditions. For a parabolic PDE one characteristic line exists, which would essentially proper get the Sonic condition. On the other hand, hyperbolic PDE has two characteristic lines, and a complete description of second order Hyperbolic PDE requires two sets of initial conditions and two sets of boundary conditions. Hyperbolic as we saw was applicable for Supersonic flow, which normally is characterized by shocks, so, essentially the characteristic lines would be characterized by the shock lines, the lines along which the shocks are present. So, that is one of the ways of looking at the characteristics.

And then of course, it needs two sets of initial conditions, and two sets of boundary conditions. So that, you need to probably impose where the shocks are going, where the shocks are anchored, so, many such issues need to be clearly defined, and of course, the initial conditions must give the Mach number - the supersonic Mach number that needs to be impose as the inflow condition. So, many of these issues need to be looked into before one, applies the appropriate PDE for finding solution to your Aerodynamic problems. We shall see later on that when it comes to Turbomachinery, the issues there are many other issues that need to be also looked into.

(Refer Slide Time: 19:23)



We were talking about the Initial and Boundary conditions. The Let us take a quick look at what these things really are, the initial condition is a dependent variable its prescribed at some initial condition, which may be at the inlet to the flow into the particular domain of interest. So, you need to prescribe whether the flow is uniform or whether it has a variable in terms of it is it is varying in a particular cross-section, whether the velocity, the temperature, the pressure, the density or the energy or enthalpy or variables or whether they are uniform that is constant at the inlet from one end of the inlet to the other. Depending on, whether you are taking a two-dimensional flow or a three-dimensional flow.

So, that needs to be prescribed as an Initial Condition of the flow going into the domain of interest. The Boundary Conditions on the other hand, are dependent variables or it is derivatives that must satisfy on the boundary of the domain of PDE. Now, if you have a domain in which the boundaries are defined. In such a case, the Boundary Conditions, some of the Boundary Conditions need to be imposed. You could have if if it is let us say for example, defining a solid wall, you may have to define that the what is the wall Boundary Condition, that means the flow across through the wall would have to be say to be 0, because you cannot have wall observing flow or flow coming out of the wall. So, that is one of the ways of defining or what the flow velocity parallel to the wall which could be 0, if it is a viscous flow that means the last layer of the flow is attach to the wall of the boundary. So, these are some of the physical conditions that would have to be appropriately prescribed as boundary condition, often the outlet flow condition may be one of the boundary conditions, sometimes the flow quality of the flow or even the quantity of the flow, may be defined in the outlet side as one of the prescribed boundary conditions.

Now, there are number of Boundary Condition that are fundamentally described, first one is called Dirichlet Boundary Condition. An Dirichlet Boundary Condition essentially is a Dependent Variable prescribed on at the boundary. For example, we have seen phi was dependent variable a potential, so those are examples of dependent variable. And that needs to be prescribed at the boundary. Neumann Boundary Condition is a Normal gradient of the dependent variable. And as I was just mentioning, you could say the normal to the velocity or the potential is specified at the certain boundary. And that needs to be specified at certain boundary condition boundaries. The Robin Boundary Condition is a linear combination of Dirichelt and Neumann Boundary Conditions. A Mixed Boundary Condition on the other hand is not same as Neumann Boundary Condition, it is some part of the boundary has Dirichlet Boundary Condition and some other part has Neumann Boundary Condition.

So, this is what sometimes you may have to consider the fact, that some part of the flow has for example, Supersonic some other flow - part of the flow may be Subsonic. There may be differences in terms of laminar flow, turbine flow, and of course many other issues that are involved. So, you may have to impose different kind of Boundary Condition under different circumstances, and you have then Mixed Boundary Condition.

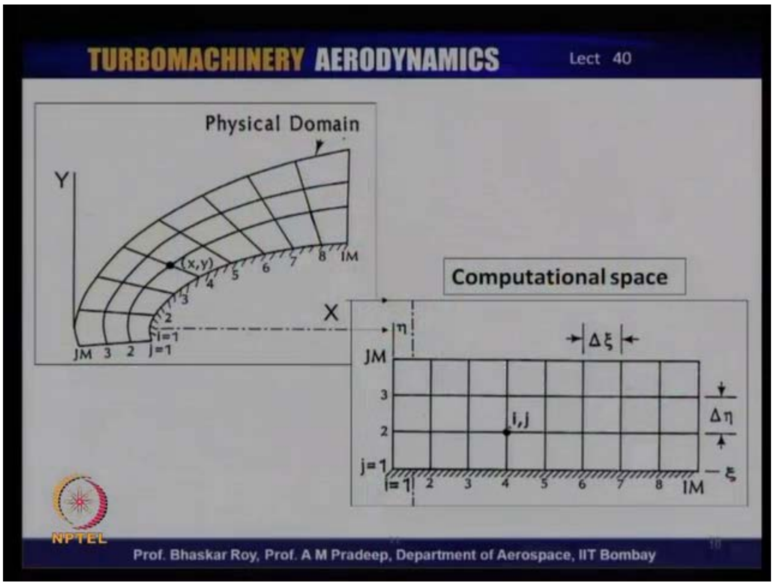
Now, the boundary conditions are applied quite often on the Body Surface. For example, R in the Far field, as I mention feel that is the Outlet flow station, let us to say let us say from Turbo machinery that outlet of a compressor or turbine at some distance would be consider the far field, then one would probably think of Symmetry as I was mentioning, you may be talking about the uniformity of the flow or non uniformity of the flow. And then you would need to probably apply exactly at the inlet flow condition or exactly at the outlet flow condition of the domain that has been prescribed for computational understanding or analysis.

So, the boundary conditions are applied under various various places, which could be a Body Surface of the body, which could be Far Field, which could be related to the Symmetry or Uniformity of the flow field or it could be related to the Inflow or Outflow condition of the domain that is been prescribed for flow dynamic analysis. So, that those are the places, where you need to apply the boundary conditions.

Now, many of these boundary conditions would have what you call physical domain. Now, physical domain is of course, the domain which you are actually trying to understand, and an analyze from you understanding an application of the Partial Differential Equations and the Aerodynamic principals. On the other hand, the computation which often is done is done in a, what is known as computational domain. So, the physical domain and computational domain may be slightly different from each other, because the computational domain facilitates computation very fast and gives you solutions at a very fast base.

The physical domain of course, is what we are indeed originally interested in. So, it becomes necessary often to transform the problem from physical domain to the computational domain through a transformation technique and such transformation techniques are been around for a long long time. And if you do that, then you are facilitating the computation in a very simple as we said it is it is basically solution of algebraic equations, and then you are computation can proceed very fast. So let us take a look at what do you mean by Physical Domain and what do you mean by Computational Domain.

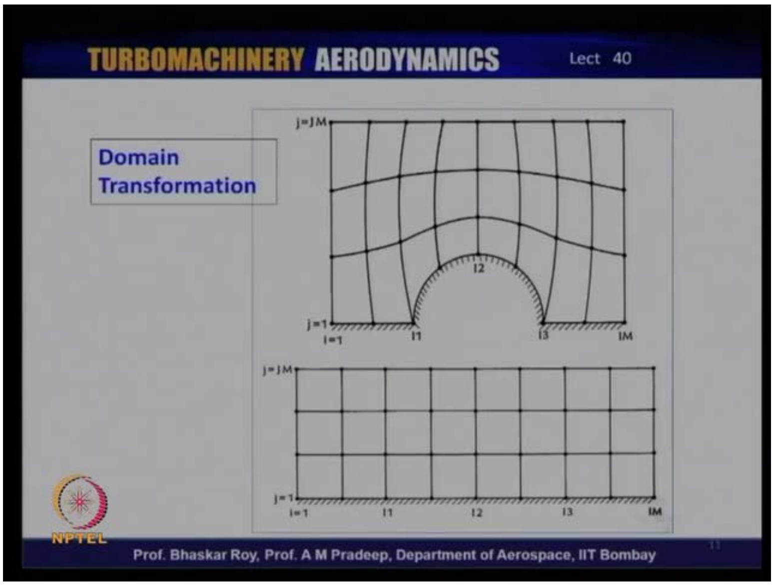
(Refer Slide Time: 26:13)



So, Physical Domain is where for example, you having a flow over this surface, which is let us say, some you know, Elliptical Surface or Pseudo Elliptical Surface. So, over this you have a flow and this flow you are trying to analyze, and find out the details of what is happening to velocity, pressure, density, etcetera, any or even temperature over this flow field; which may be let us say part of an aerofoil or part of a blade. For example, of compressor or turbine you know you can notice that it could be somewhere near by leading edge.

Now, this physical domain would need to be transform into this computational domain which is straight forward domain. In which each of these, you know, grids or or domains, the small part of the domain let us say, would have to be transform into this straight forward rectangular domains. And as a result of this, the computational solution in this computational space would proceed very fast. So, there are transformation techniques by which this transformation is often done from physical domain to a computational space to facilitate computational to proceed with the computation.

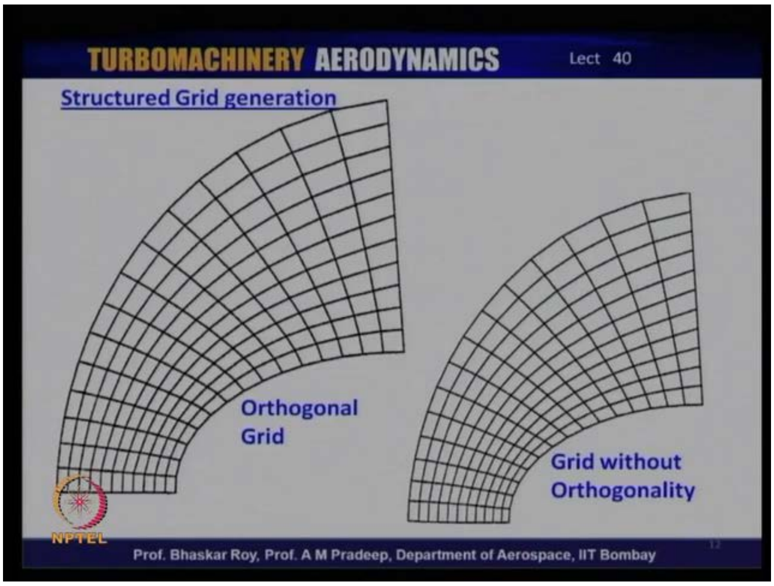
(Refer Slide Time: 27:41)



There is an other kind of domain transformation again; where let us say, this is the physical domain physical domain. In which, you have a flow around this which could be a part of a sphere or circle or cylinder, and then the flow is having this kind of a characteristic. This needs to be then converted to this computational space in which the characteristics can then be found, and then again from point to point basis they have to be transformed back or transform back into the physical domain to get you a physical solution. So, you can get your computational done in the computational space, you still need to transform it back into the physical domain to get your physical solution. So, quite often, in in terms of computational polence, the computational may converge or may give you good solution in the physical domain, but you may have difficulty converting back into the physical domain, and then the computational would have to be considered has not fully converged that means a full solution has not been obtained.

So, there are certain some issues where transforming it back could be a little problem. Of course, transforming it forward from physical to computational domain is indeed and issue or a little problem which needs to be solved through certain transformation techniques. So, both forward and backward transformation needs to be completed before you can complain, before you can claim that you have a solution that is applicable to the physical domain.

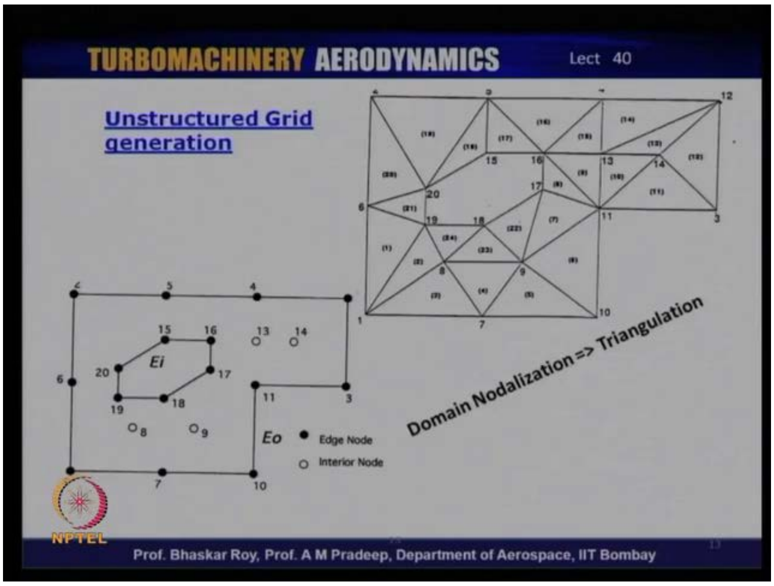
(Refer Slide Time: 29:34)



Now, to create this computational space quite often, you need to create as you have seen measures are what is simply call grid. Now, there are essentially two kinds of grid that are you being use these days; one is called the Structured Grid and other we will see in a little while which is called Unstructured Grid. Now, the Structured Grid has you may have added some already introduction in the earlier lectures. Essentially, has very simple, you know, Grid Structure in which one way of doing the Structured Grid is to have Orthogonal Grid. So, if you look at this domain over here which is been broken up into so many, you know grids, and each of this junctions is of course called the node. And at the node in this Orthogonal Grid, the two surfaces or two lines are essentially almost parallel to perpendicular to each other.

So, this perpendicularity or orthogonality makes it an orthogonal grid. So, even when you are creating these grids; one has to be careful that at each of these points you have maintained orthogonality. The other way of doing it is where grid is created without orthogonality, which means these lines are actually non orthogonal to each other, and they can indeed then be retained as some kind of a curve line. So, orthogonality is something, which can be maintain which is orthogonal grid or grid. The modern way doing it could be where you have grid without orthogonality and those are sometimes referd to as quasi orthogonal grids. We shall have an example of this, when we can get into solution of Turbo machinery in the blades in a little while from now.

(Refer Slide Time: 31:44)

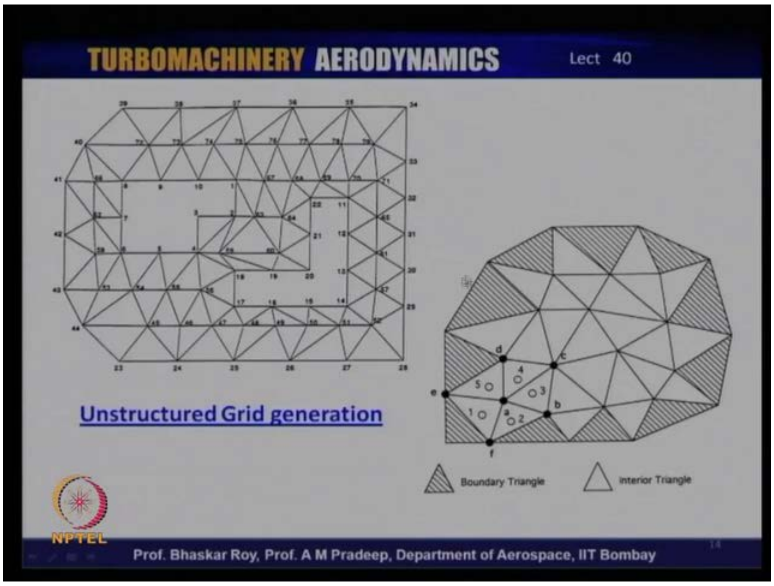


The Unstructured Grid, on the other hands, starts off with certain identification of the boundary, and then the inside space which you need to have solution to, and then you identify the nodes on which at the solution need to be definitely found in terms of let us say velocity and pressure and other flow parameters. You need though solutions at those points, and of course, what probablic are known on the boundary. On the other hand, what needs to be done is this Domain Nodalization which has been done.

So, you need to first nodalize them create the nodes at which you want the solutions definitely. And then you resort to what is known as triangulation. So, if you connect these nodes, you start getting triangles. So, once you start getting triangles, the solution or the computation can proceed from one node two another. And as a result of which all of them now are sort of connected through triangles.

So, this triangulation then becomes necessary step to all grid generation. And has we see more, and more and as we have done in the earlier two lectures grid generation is an important, and very crucial and critical step in the process of Computational Flow Dynamics. You need to create grid that appropriate to your physical problem, and that is the first problem or the first issue that CFD people would have to sort out, what kind of grid needs to be created to find a solution. So, grid is essentially towards finding a solution, towards finding the solution to the Partial Differential Equations. So, creating grid-appropriate grid is an important issue in finding CFD solutions.

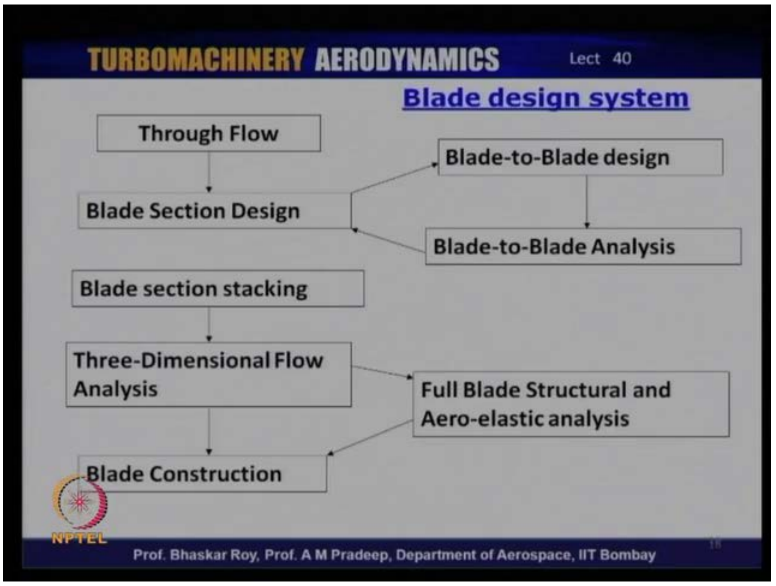
(Refer Slide Time: 33:49)



Here, you can see little more involved three-dimensional Unstructured Grid Generation. In which triangulation has been completed and now you can see, you have triangles on the boundary. So you have the shaded ones which are triangles on the boundary; and these white triangles are the interior triangles in the interior of the body. So, we are talking about a three-dimensional body, inside of which, we have the triangles and outside on the surface we have the boundary triangles. So, when you have a three - dimensional space or three-dimensional body, over which, let us say flow needs to be captured through various CFD techniques, you need to resort to this kind of Grid Generation.

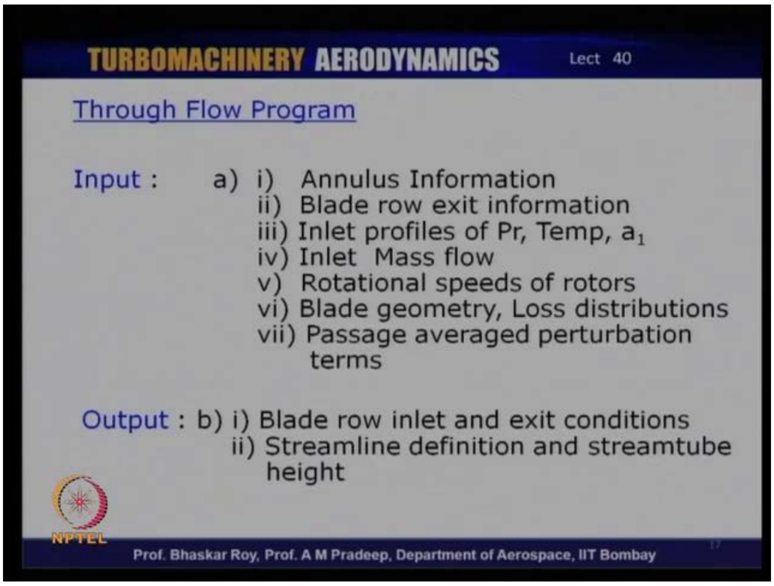
So, Grid Generation as I mentioned and as you have done in the earlier lectures, is an important issue, that needs to be sorted out, many of the CFD packages that are available these days, do have certain some time they have certain automatic grid generation facilitated, but sometimes you have to really set down, and create a grid appropriate, and which is not easy, which often takes quite a lot of time, before you have a grid that gives you; a good solution that is useful to you. So, grid generation through some of the things that we have been talking about; actually, is a it is a time consuming issue, some of the packages facilitated automatic grid generation, but then you are then you are using something that is automated and develops, and you have no control over it. So, many of the CFD people would like to have control over the grid generation, and they like to create grid of their own, in which case, some of things that we are talking about would indeed need to be done in a more elaborate and very deliberate manner. So that, you have compressive grid, which in compressors you entire flow domain, in which you would need to have your solution of your flow dynamic problems; so, let us take a look at some of the issues related to Turbo machinery.

(Refer Slide Time: 36:12)



Now, CFD in blade design essentially proceeds through number of steps. The Blade Design System actually, as we have done earlier, you have a Through Flow technique; and then you resort to the Blade Section Designs, so section by section, that is aerofoil by aerofoil, you need to develop Blade Sections, which are indeed aerofoils; and then you resort to Blade-to-Blade design or Blade-to-Blade Analysis. We will talk about what is this Blade-to-Blade Analysis? And then you have Blade sections stacking. So, all these section that you have created need to be stacked in a certain manner; and some of the stacking techniques are changing in the process of stacking, people are applying sweep and dihedral to blade shapes, so some of those things we have talked about earlier, and that is what stacking essentially means; and then finally, the stacked full-fledged blade would have to be subject to three-dimensional flow analysis. We will simply try to show you today, what the three-dimensional flow analysis would indeed, mean or the beginning of the three-dimensional flow analysis, which would need to be later on supplemented with Full fledged Blade Structural Analysis and Aero-elastic analysis, which is be outside the preview of this course, on this lecture series, and but only after all that is done, you go for Blade Making or Blade Construction. So, Blade Design is something, in which lot of things need to be done in a very step by step manner, very patiently, and you need to go step by step, so you need to create Blade Sections to begin with. So, let us take a look at, what it means to create a blade using CFD along the way.

(Refer Slide Time: 38:12)

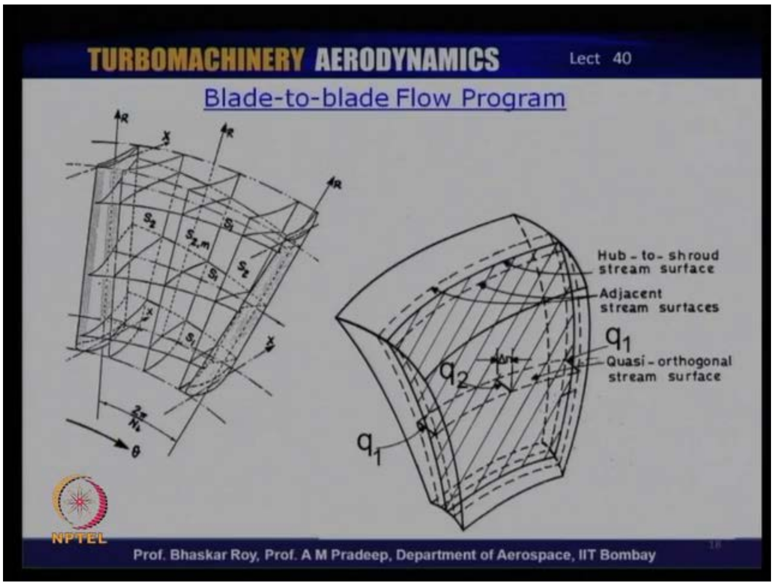


Now Through Flow Program, which is what you would probably need to look at first, to know, what kind of flow that you are creating, you need to have the Annulus Information that means to need to have the tip diameter, the hub diameter decided already, and that decision has to come from various design parameters that we have talked about earlier. And then you need to a probably put in the Blade row exit information, sometime this exit information are imposed on the Blade Design that means, the exit would have to be axial or the exit would have to be at a certain angle or the exit would have to be certain angle have to be turbine, for example, at a certain temperature or the exit would have to be under certain flow uniformity condition, those things would need to been imposed, and as you can as you know input dynamics, they would be the outflow conditions or essentially boundary conditions at the exit flow field; then of course, you need to provide the Inlet profiles of pressure temperature, and the flow condition in terms of what should be the sonic velocity; So, those need to be supplied to the designer.

And then, the Inlet Mass flow; now, Turbo machinery is always, as we have seen design for a specific mass flow. If it is air as the flow medium, then the air mass flow has to be very accurately determined apriori, before that the design is initiated. So, mass flow of the air or any flow it through the Turbo machinery needs to be decided apriori before the design is on, and then you need to decide the Rotational speeds of the rotors; this needs to be decided apriori, otherwise of course, you cannot have a design, and then the blade geometry the loss distributions would need to be created through some loss models, which are sometimes empirical, some empirical loss models; the blade geometry needs to be created with the help of the design methodology that we have done in the earlier lectures.

So, you need to have a blade geometry to go into CFD; this is something we have mentioned when we were discussing blade design that blade design essentially creates a first cut geometry, as good as possible a first cut geometry, which is then subject to CFD analysis; and then the Passage averaged perturbation terms, there are all lot of many other issues here, the perturbation terms, the periodicity, many of these issues would have to be defined, before you get into the through flow program. The output from this would be in the form of a Blade row inlet and exit conditions, you get the details of those things, and then the Streamlines are well defined, and the stream tubes are created, and all the details of the stream path are captured through the CFD solutions. So, these are the input and output of typical through flow program that you would need to have with you, when you get into to the design mode.

(Refer Slide Time: 41:34)

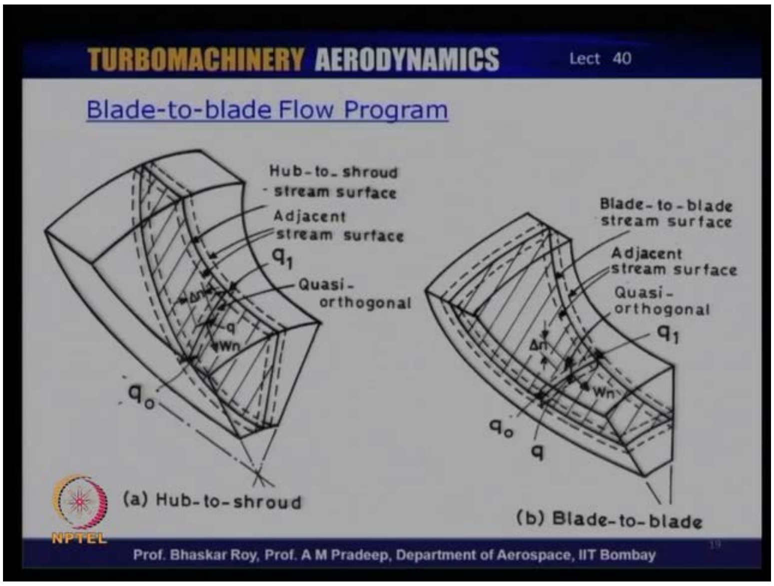


Now, we will take a look at, what is meant by Blade-to-blade flow program? See, you can see here, there are two blades: one over here, and other over here, and what it means is that you need to capture the entire flow domain from this blade to that blade, and which essentially means that also of course, you have a hub and you have the shroud or casing. So, this is your flow domain and on one side, you have one blade and other side you exactly a similar blade; they are same shape. On one side, you have the hub; on the upper side, you have the shroud or the casing. So, this is your captured flow domain, in which you need to find your detail aero dynamic solutions.

So, Blade-to-blade flow program essentially, tries to move from one blade surface to the other; that is one way, that is one may call them is one surface and then of course, you are S 2 surfaces, which move you from hub to the tip of the blade. So, you can traverse cross section by cross section from one blade to the other; and then, you can traverse from hub to tip again cross section by cross section, each cross section is from one blade to other, here, each cross section is from hub to casing. So, this is one cross section from hub to casing; and this is another cross section from hub to casing; the third cross section from hub to casing. So, you are moving from one blade to another, you have to remember of course, that this surface of this blade; and the phasing surface of this blade are dissimilar. So, when you move from this surface, finally approach the other surface. The surface shape would be changing all the time similarly, when you move from hub to the casing, the hub has one curvature the casing of course, has another curvature. So, as you move from hub to the casing cross section by cross section, the curvature, the radius of curvature would indeed change. So, this is the Blade-to-blade flow programs that quiet often, the CFD people use in Turbo machinery.

This is the space, which we are talking about the typical space within two blades, and you have the two surfaces defined over here the q 1 and q 2 are the quasi orthogonal stream surfaces; and this is what you can see that the two surfaces are not orthogonal, they are quasi orthogonal; this is something which we indicated a little while back. So, one is the hub to shroud stream surface. All the way from here to here, the one which is hatched; and then it has a certain adjacent stream surfaces within, which a flow solution is to be found; and then doing that you move from all the way from this surface to that surface from one blade to the other; other way is you have this surface, which is quasi orthogonal; and that has certain adjacent stream surfaces, and within that flow domain you move from hub to the tip. So, these are the methods by which you can capture all that is happening inside this that means all that is happening inside the blades or blade passage between two blades of compressor or turbine.

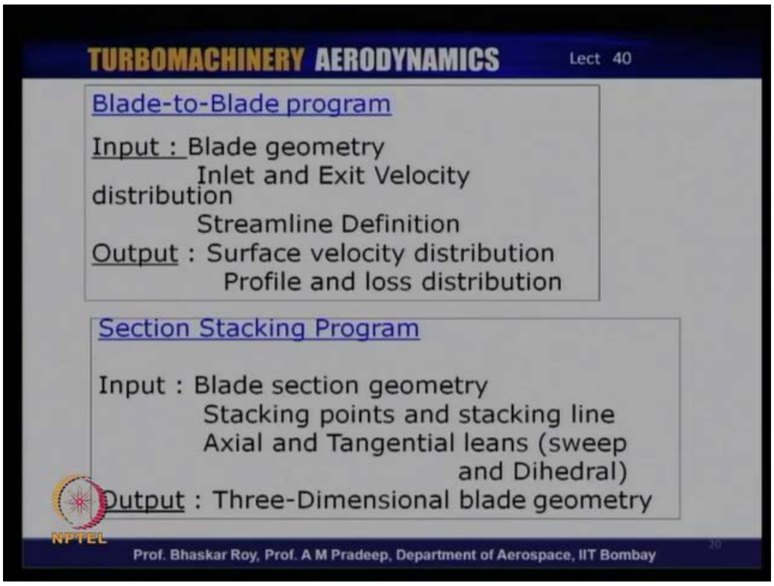
(Refer Slide Time: 45:10)



This is another Blade-to-blade flow program, in which you can capture flow domain, again from this surface to that surface; this is a surface, in which again you have the quasi orthogonal, and the the middle of which is the stream surface, and then this is a Hub-to-shroud stream surface. So, this is near the hub; that is near the shroud; and as you can very well might have already guessed, this is a flow domain that is typical of a centrifugal machine, the earlier one you were looking at typical of actual flow machine compressors; the actual flow compressors and turbines; this is typical of centrifugal machine that we have done the compressors; and the turbines.

So, again this is another kind of surface, which is Blade-to-blade, so that is the surface is from one vane to the another in centrifugal machines we called them vanes, so this is from one vanes to another; and so you move from hub to shroud surface by surface and of course, you have the surface defined which is quasi orthogonal to the other surface. So, you can again cover the entire space between two vanes, all the way from hub to shroud by moving either along this surface or moving along that surface; and as result you captured the entire flow domain inside two vanes. So, this is what is simply referred to as Blade-to-blade flow program.

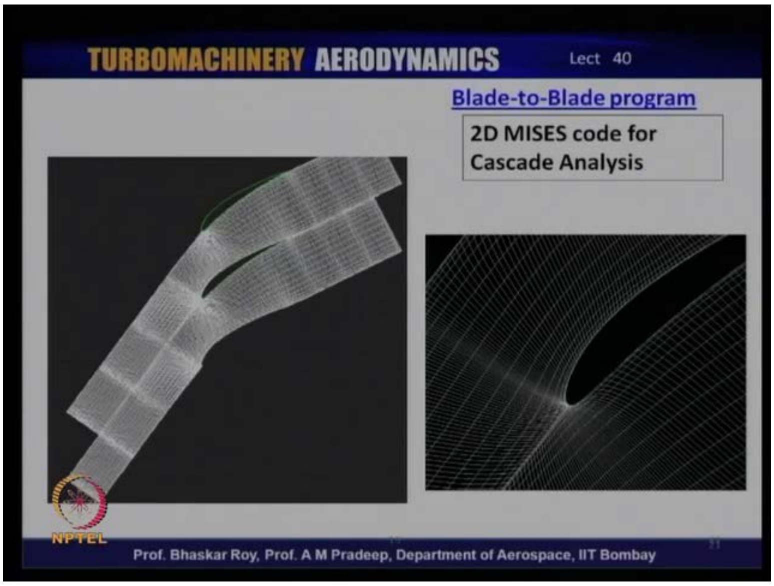
(Refer Slide Time: 46:45)



So, here what what all the things, you need the Blade geometry; you need the Inlet and Exit flow distribution, and you need the Streamline definitions. The Output of which of course, would give you the Surface Velocity Distribution, the pressure, and the loss distribution. These are the things of course, which you need, the section stacking, which is very important in actual flow machine, you need to define or have decided upon the aerofoils, the Blade Sections - the blade section Geometry, which are as I mentioned aerofoils, then the Stacking points and the Stacking line. Most of the conventional design has been whether stacking line is radial. So, all the blades are aerofoils are stack radially, as we have discussed before, but many of the modern blades are being stacked non radially; in an attempt to give Sweep or Dihedral to the blade shape, we had a few look at few of these blade shapes. So, that is how stacking is being done in some of the modern blades. And then these non-linear stacking lines then provide Sweep and Dihedral. So, the exact amount of Axial and Tangential leans would have to be appropriately decided and prescribed to apply certain amount of Sweep or certain amount of Dihedral, what you get out of these is three-dimensional blade geometry.

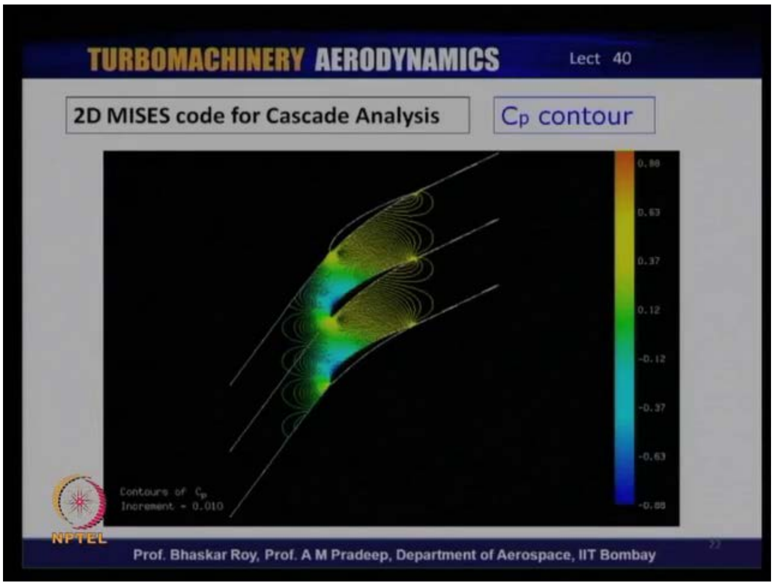
So, the Section Stacking Program, which essentially is a geometric modeling, gives you the three-dimensional blade geometry. So, the Blade-to-blade flow program gives you the Surface velocity distributions, the C p distributions, for example, on an aerofoil and the profile and the loss distributions over these various blade shapes; two-dimensional or three-dimensional loss distributions which would go into finally, finding the compressor or turbine efficiency and their exact performance parameters.

(Refer Slide Time: 49:00)



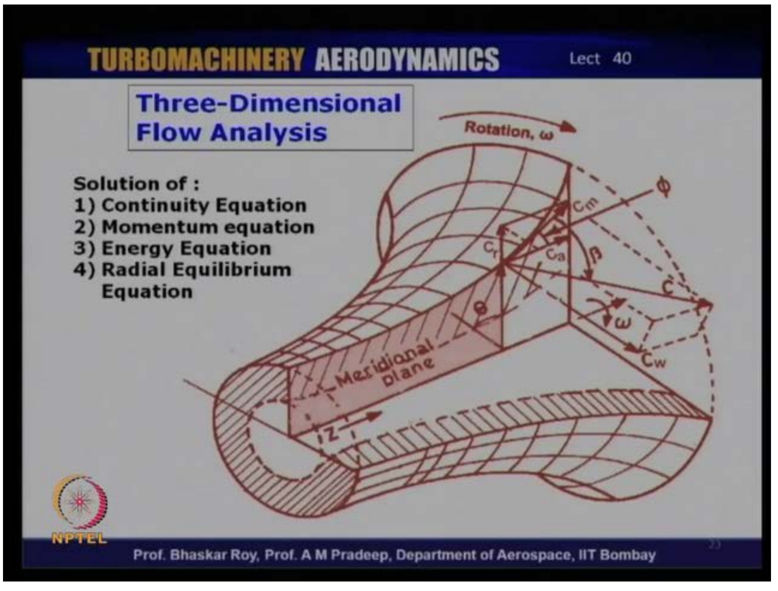
Let us take a look at, some of the solution that you can get typically get out of dimensional solution of a Blade-to-blade program. Here, the domain was all the way from here, all the way this is the whole domain inside, which you have two blades in a cascade formation, such that the flow goes in in a prescribe manner; and then those over this through this two passages subtended between three blades, one, two, three; and you can see the one surface of one of the blades. So, two passages what is being analyzed, and these two passages identical to each other; and you have to maintain periodicity of the flow both in front, as well as at the exit. If you can do that you get a solution of as we just stayed at what is happening over the blade surface in terms of velocity; and pressure; and you can get the C p distribution this is a close up of, what is happening the grid generation, the grid that have been created to capture exactly, what is happening in front of leading edge of an aerofoil; you need very fine grids around here near the surface, to capture all the things that are happening around the blade surface, around the leading edge. So, this grid generation as I mentioned earlier, is something that needs to be understood; and applied very judiciously to capture all the details that you need to know to get on with your blade design.

(Refer Slide Time: 50:36)



This is an typical output which you finally get of that particular analysis, and here you can see the C p control in terms of the color code that is available; and you can see the pressure of the C p distribution over the entire flow domain, as was described in the last slide. So, this is the out kind of output, you would get, if you do everything properly in a described or prescribed flow domain. This was a two-dimensional cascade flow analysis.

(Refer Slide Time: 51:12)



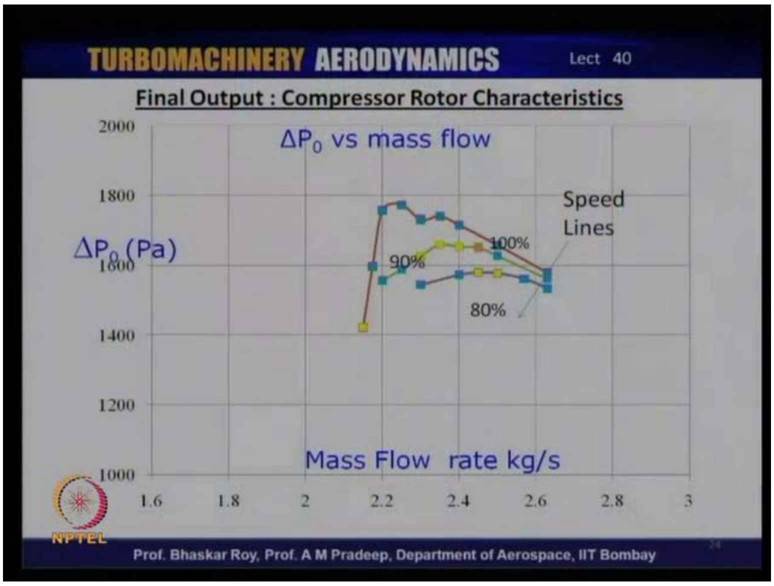
If on the other hand, you want to do a three-dimensional flow analysis, this is what you would probably need to do, you would of course, be trying to solve the continuity equation, the momentum equation, the energy equation, and in turbo machinery; this is very important, you need to also bring in radial equilibrium equation or the radial balance of forces that we have done earlier in the earlier lectures. So, this is the flow kind of domain, he would have for the three-dimensional flow analysis, in which you would need to solve all these equations; and find out what is happening this is the meridianal plane that we have described before consisting of the actual direction, and the radial direction at any given cross sections. So, it consist of the actual and the radial access and this is where you have all the velocities that we have described before the whirl component C w; the radial component, which the flow often acquires as it gets inside the Turbo machinery C r, the actual velocity C a, and the meridianal components C m, which is the combination of or vector product of C a and C r. So, that is a meridianal velocity, so this is in rotation with angle of velocity omega; and C is the three-dimensional result in velocity; this flow is experiencing phi is the meridianal flow angle.

So these are the prescriptions with which you proceed with your three-dimensional flow analysis. So, it is a long drawn out process quite often in in we have seen you may have a single stage, you have a multi stage, and then the flow solution proceeds through rotor stator rotor stator, and it is a multi stage flow solution which often is a very as I mentioned long drawn out affair. So, many of these things require a lot of time, lot of computation time, so computation in Turbo machinery is a very time consuming affair compare to simple solution over an aerofoil.

So, some of these things need to be done in a step by step manner; and of course, more you know about it you can do it appropriately, correctly right from the beginning for example, the grid generation the choice of the grids and many other issues that have been talked about in your last two lecturers on CFD. So, all that needs to put together to find a good solution for a particular Turbo machinery. In Turbo machinery everything is case by case you have to solve every there is nothing like a general solution, every compressor; every turbine centrifugal radial would have to be solved on a case by case basis. If the flow conditions change you have to find another solution; if the inlet flow condition of change you have to find another solution; if the outlet outflow condition is changed the pressure field or temperature field is change you have to find another solution. So, many of these issues require fresh computation and you have to get in to another round of computational exercise, to get a find another solution. Some, of these are related to fine tuning if you change the blade geometry just a little you think it is just a little but, it requires a full pressure computation all over again.

So, many of these things would have to be understood, as you go along apply the principles of Turbo machinery that we have done, and apply the principles of CFD that we have just introduce to you and you would be able to then get together solution and a blade design that is good efficient design; and gives the performance that is wanted by the Turbo machinery engineer in the first place, which can be then be send for fabrication and construction. So, these are the issues that are related to CFD.

(Refer Slide Time: 55:37)



At the end of CFD, he would for example, in a compressor he would need to produce a compressor characteristic map like this out of CFD. So, CFD would give you a compressor map, which is delta P 0 verses mass flow typical compressor map that you have done before and such a map can be produced by CFD before you go for rig testing. So, these are the speed lines and at various speed lines, you get various maps, various characteristic lines of the compressor, to tell you how good this compressor is, for example, where the stall is going to occur it gives you an indication that this is where the stall is going occur at this mass flow; and at at which this is the kind of pressure after which the compressor is stalled, so this is something which you can get out of a through flow CFD program; and it tells you the characteristics of the compressor as a first cut output of your CFD analysis; and then you can decide whether this design is good or bad or whether you like to do differently before you go for a very costly business of making the compressor or turbine and going for rig testing, because those are very costly things of course, the computation of Turbo machinery also as I mentioned in terms of computational cost is also quite often quite costly but, it is much faster than going for very time consuming and laborious rig testing.

So, this is how you you CFD in making Turbo machinery blades, which are modern compressor; and turbine blades; many of the modern turbine compressor blades have indeed been produced with the help of CFD, we have tried to bring to you a number of issues related to Turbo machinery, the compressors, the turbines, the centrifugal, the axials and we have tried to give you some idea about the fundamental principles of aerodynamics. We have not got into the structural issues; that is a separate issue all together, you probably have to deal with it in a separate course.

We have deal with only the aerodynamics issues in this course of Turbo machinery aerodynamics; and at the end we have try to bring together into your attention the use of Computational Fluid Dynamics in design and analysis of Turbo machinery. On behalf of professor A.M Pradeep and myself, I would like to thank you for participating in this course, I hope that we have been able to bring to you the joys and the challenges of this course, and hope that some of you would spend some time learning it more, and may be choosing it as your carrier, in your future carrier after the your programs are over. Thank you very much again on behalf of professor Pradeep and myself.