Turbomachinery Aerodynamics

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

CFD for Turbomachinery: Grid Generation, Boundary Conditions for Flow Analysis

Lecture No. # 38

Welcome to lecture number 38 of this lecture series on turbo machinery aerodynamics. We have come towards the end of this particular lecture series, we have three more lectures left and we have decided that as we promised in the first lecture that we are going devote these three lectures towards discussion on some aspects of CFD or Computational Fluid Dynamics specific towards turbo machine flows. We are assuming that you are aware of some of the basics of CFD and that you have had a chance to understand the fundamental aspects involved in CFD. So, we are assuming that this background information is available with you and with that assumption we are going to discuss some of the aspects which hold the key towards using CFD as tool for design as well as analysis of turbo machinery flows.

CFD as we know is relatively young compared to the other two methods of analysis which have been existing for a very long time now. The other two methods being the theoretical analysis, methodology as well as the experimental method which they of course, these have been around for several years now, and therefore CFD in comparison to these methods have relatively very short span I would say may be last 25 to 30 years also. So, it is in the last 25 to 30 years also, that there has been a tremendous development in techniques or design analysis methods using a third approach which we now know as Computational Fluid Dynamics or CFD.

So, CFD is basically trying to solve the governing equations of a flow using a computing technique using numerical techniques and that requires that we identify the domain of interest and then decide which are the points on this flow that we would like to solve and then that is considered to be taken as a representation of the flow field itself. So, depending upon the number of points that we choose to analyze you can get better and better solutions depending of course, upon a variety of other parameters like the solver being used and so on.

So, CFD though it is considered as a very powerful tool in the design analysis and optimization loop which is a common loop that is used in any design exercise that it is taken up, that there is a preliminary design then its goes to as 1-D analysis and then 2-D and 3-D analysis and then it goes to an an optimization routine and then finally a CFD detail CFD analysis and then this loop is kind of if required from the CFD analysis to be find that the performance is not what we had intended then the designers have to come back to some of the intermediate steps to correct those issues and try and achieve the performance that the particular design exercise was intended for .

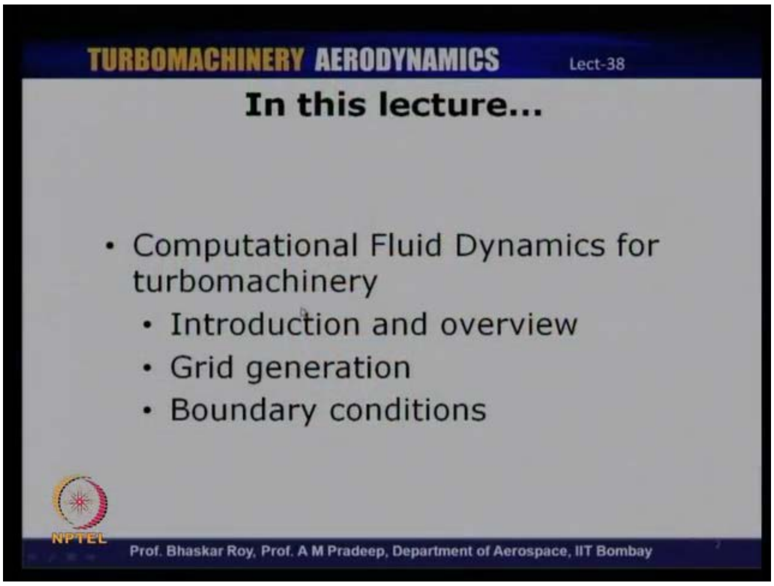
Traditionally these have always required the design analysis and optimization loop always required an experimental validation of the design itself. This is partly being replaced by Computational Fluid Dynamics, but as has been as was initially thought that CFD is going to replace all numerical, all theoretical and experimental techniques that is quite not true at least at that moment and few years from now that I can for see, that experimental methods as well as analytical or theoretical methods will continue to guide us or be the other two distinct analysis tool that will continue to exist along with CFD.

Of course, the importance and relevance of CFD is continuously growing with more and more modifications and refinements that CFD tools have had in the last several years and with the computing power that is available which is also increasing at an at a very fast rate, CFD is definitely likely to be a very powerful tool which will be used and it is continuously even now used and will continue to be used even more in design analysis in general.

But our interest as in this course is on turbo machines and CFD of course, has been used in turbo machine design analysis and optimization cycle, but there are certain very key challenges which are still which which still need to be resolved. So, that CFD can be you know taken as a very standard technique of design. Even though it is, but there are certain limitations which are which is what we shall be discussing in today’s lecture and possibly in next lecture as well.

We will continue with some of the issues associated with CFD and those which are currently under revision in the sense that they need to be revised and better methods of estimation of certain aspects need to be developed which hopefully will happen in the coming years.

(Refer Slide Time: 06:03)



Today's lecture we are going to devote towards basically three aspects. CFD I will give a general introduction which I assume that you already have, but I will nevertheless give you an introduction and overview of CFD then we shall talk about grid generation and boundary conditions.

Grid generation is is still an issue with reference to CFD and of course, grid generation in general if you talk to people who work in CFD they would admit that grid generation continuous to be relatively challenging aspect because that something which is very much dependent on the geometry that we are trying to solve. So, the more complex and intricate the geometry the more difficult it is to generate grids for a particular geometry. In fact, considerable amount of time is usually required for generating the grid for a very complex geometry. For example, if one has to generate a geometry for a very complicated geometry let us say like combustion chamber including all the holds and cooling holds which are present on the combustion chamber or for that matter even a turbine blade which has all the cooling holds and which need to be simulated using CFD.

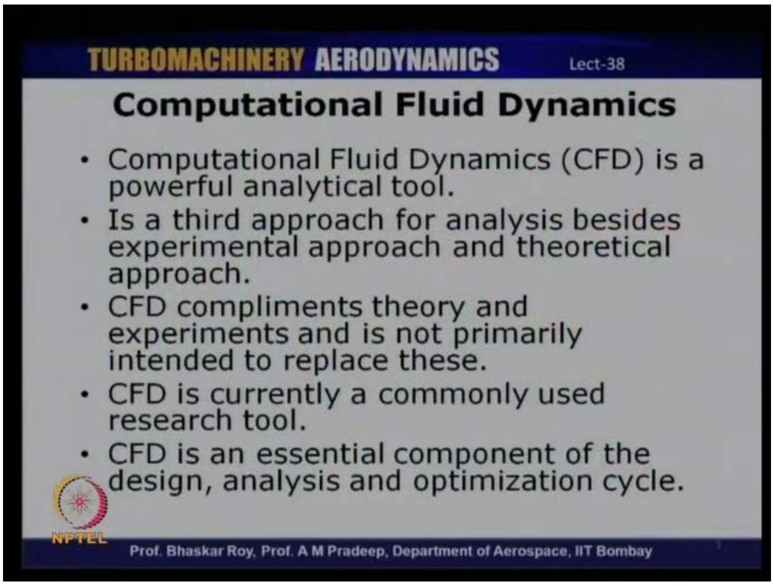
So, generating grids for such geometries is extremely complicated and a tremendous amount of time is required for developing grids which can accurately predict the performance and and people all over the world researches are trying to develop automated tools which can be used for generating a grid which in some form exist for relatively simpler geometries, but for very complex geometries like with turbine blade cooling holds and combustion chambers and all that it becomes a later tricky and we did not really have an automated grid generation tool which can faithfully grid the such very intricate geometries and help us in the analysis.

So, CFD as I was saying is considered is now considered a standard third approach which comes in the design analysis optimization cycle. The other two approaches as I mentioned being the experimental approach as well as the theoretical approach. CFD definitely is a third approach and is increasingly being used by designers in their design analysis cycle.

CFD has always been intended and will continue to be intended to complement theory and experiments. It is not meant to replace either of them will which is unfortunately very rather large misconception amongst people that CFD is something which can replace experiments and theory and I do not think that is going to happen anywhere in the future. CFD is going to compliment theory and experiments as third approach and would of course, be increasingly used by designers all up the world.

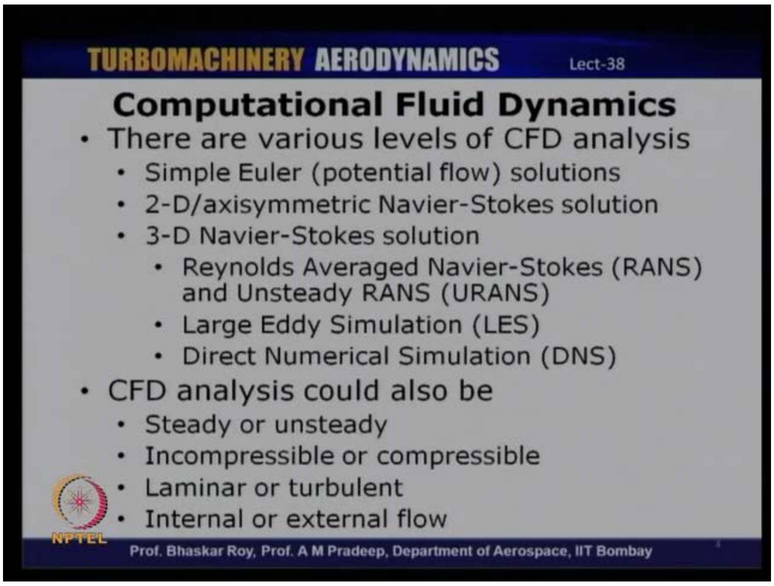
And CFD is also very common research tool in the sense that there is lot of design exercises which involve lot of research to be carried out on let us say optimization or a new design to be developed through certain modifications of the shape or any other method that involves a lot of research exercise to be carried out. CFD is definitely a very strong contender for one of the research tools which can be used for such an analysis.

(Refer Slide Time: 09:41)



So, CFD is to summarize a few points and I was talking about CFD; obviously, is a very powerful analytic tool and is a third approach for analysis besides experiments and theory. CFD compliments very well theory and experiments and is primarily not intended to replace any of these and off let CFD is very commonly used to research tool and it is definitely recognized as a dependable research tool for a variety of flow applications accepting a few cases where CFD is still struggling to kind of predict the performance very well. For example, in compressor stall surge prediction that is something which CFD is not really in a position to predict well and that is where some of these experimental or analytical tools will still need to be used for such complex flow scenarios.

(Refer Slide Time: 10:37)



Now, let me quickly take you to the different methods or levels of CFD analysis that could be carried out. So, one could carry out very simple fast simulations or one could carry out very detailed and analysis of a particular flow field that it trying to simulate and that depends upon the requirement, whether we really need it is very such high and computations to be carried out which obviously requires a lot of time as well as effort and, and therefore money or can simple calculations using CFD help us in understanding the the design methodology and whether the basic design works or not. So, based on this one could either have very simple Euler based solution which is a potential flow solution which could again be 2-D or 3-D or one could go for one level higher we could go to 2-Dimensional or axisymmetric Navier-Stokes solution with of course, certain approximations or one could go for a 3-D Navier-Stokes solution, one could either have a Reynolds Averaged Navier-Stokes - RANS as it is called that is truncated form of Navier-Stokes equation using Reynolds Averaging or one could go for an Unsteady RANS that is also called URANS - Unsteady Reynolds Averaged Navier-Stokes.

Another level higher is what is known as Large Eddy Simulation which is something which is much more involved in complicated than URANS or RANS. Large Eddy Simulation involves simulating the larger Eddies and computing the smaller Eddies directly. Larger Eddy Simulation; obviously, requires much higher computational power as compare to RANS or for that matter any other Navier any other form of solution and the ultimate aim of course, is to get what is known as a direct numerical simulations or

DNS. Direct Numerical Simulation is something which is used well which could be eventually I would say few years from now may be could be used for complex flow scenarios like the turbo machines. Currently this is simply being respective to very simple flow fields like flow path circular cylinders and flow passes an airfoil and so on.

DNS involves use of the Navier-Stokes the 3-D Navier-Stokes equation in its original form without any approximations. Unlike in RANS where there are certain approximations for example, turbulence is modeled in in RANS, DNS does not require any turbulence modeling, and therefore that is considered to be the most accurate most possible accurate numerical solution of a flow field, but; obviously, this requires a huge amount of memory and also DNS is directly a function of the Reynolds number. Higher the Reynolds number the more are the number of cells that will be required for for us to develop a DNS solution. DNS in fact, is proportional to Reynolds number square which means if we are looking at a Reynolds number, the number of nodes required for DNS is square of the Reynolds number itself almost the square of the Reynolds number.

So, if you are looking at a Reynolds number which is let us say in typical turbo turbine or compressor flows could be easily in ten to the power 5 or 6. So, square of that is 10 rise to 10 or 10 rise to 12. That is the amount of nodes or elements or discretized elements which we will we required for us to develop DNS solutions.

Now, one could also have CFD analysis which could be either steady or unsteady as I was saying, one could have RANS solutions or one could have unsteady RANS solutions. One may also have is the flow requirement is such that the Mach numbers are very low. One might stick to an incompressible CFD analysis or if one is dealing with higher Mach numbers then that is basically a compressible solution that we need to look at.

If the Reynolds numbers are very low it could be either laminar flow solution or if it is high Reynolds numbers flow, then we need to go a turbulent flow simulation. One may be also dealing with internal flow or external flow. In turbo machines, generally the flow field that we are trying to simulate are internal flows, but if you looking at let us say a simple airfoil or a blade shape without considering the casing and all that then that could be considered like an external flow stimulation, but in general turbo machine flows are internal flows simulations that we carrying out.

So, these are different methodologies that are available for a designer to choose from and try to apply some of these methods in or incorporate these methods in his design exercise and depending upon the level of accuracy that is required from the simulations the designer may choose for either a simple 2-D Euler solution or a 2-D Navier-Stoke solution or one could go for a 3-D RANS solutions and possibly when an LES depending upon the computer power that is available, but not a really a DNS at this moment we are not really at a stage where we can use DNS for full a scale, let us say compressor flow simulation or even a turbine flow simulation we are not really up to that level, but hopefully in the next few years we should be able to develop techniques which can be or develop computing power which can be used for using DNS for our numerical simulations.

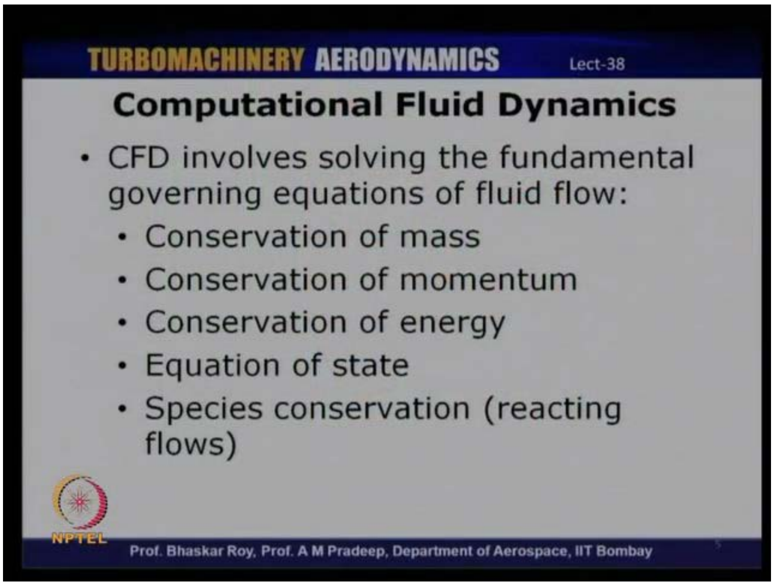
So, DNS is likely to be depending upon other developments which might take place and which probably requires much less computing power. DNS could possibly be the ultimate aim of CFD user or a CFD researcher where one can use DNS in flow field simulations and capture the entire range of scales that are there in a turbulent flow. For example, if you are aware of turbulence and turbulent flow situations then we know that in turbulent flow energy dissipation is taking place through Eddies which are which which break down into smaller Eddies and eventually dissipates the energy and the smallest scale through which energy is dissipated is known as the kolmogorov scale and it has scales in length, time as well as velocity.

So, these are there are several ranges of these scales which are there through which energy is dissipated and in large Eddy simulation, we try to compute this smaller Eddies properly and simulate using certain approximation the larger Eddies and that is why it is called large Eddy simulation and since we are simulating larger Eddies there is and there is scope for some approximation that is coming in which is why earlier results though or much better than RANS solutions, they are still not the final or the correct solutions.

Well it is correct in the sense, that it depends upon the level of accuracy are we looking at where as if you look at DNS there are no such simulation or approximations that are taking place it is computely it is completely simulating or calculating all these scales present in a turbulent flow which means that to be able to capture the smallest scale from smallest scale to the largest scale that many number of grids or nodes are required where all the governing equations can be solved and that is why it is called direct numerical simulations which does not involve any approximation. Currently the most commonly used 3-D analysis tool is the RANS or Reynolds Averaged either in the steady mode or Unsteady RANS that is URANS.

But the only issue, one of the issues which RANS or URANS has is in the computation of turbulence because we as such at the moment do not really have turbulence model or a model which can simulate the turbulence in the right way. There are several turbulence models which are available and designer has to choose among these set of turbulence models which are available depending upon the applications. So, there is no model which can be set to be universal and can be used in all applications their application depended and that is one of the limitations that modern day CFD has when it comes to computing turbulent flow. Now, I mentioned that the governing equations are the once which are being computed or solved over a flow field.

(Refer Slide Time: 20:04)



Let us take a quick look at what are the different governing equations that are been solved in a particular in a typical CFD simulation. I am sure you would be aware of this, but this is just to recap fundamentals.

So, CFD basically involves solving the governing equations of fluid flow conservation of mass, conservation of momentum, conservation of energy. One would also be using equation of state and the species conservation in case it is a reacting flow. For example, in a turbine flow one as there are hot gases which are present in a turbine flow and so, one may also be required to use the species conservation equation because one would like to ensure that the species in terms of all the constituents of the hot gases are conserved as it passes through the turbine.

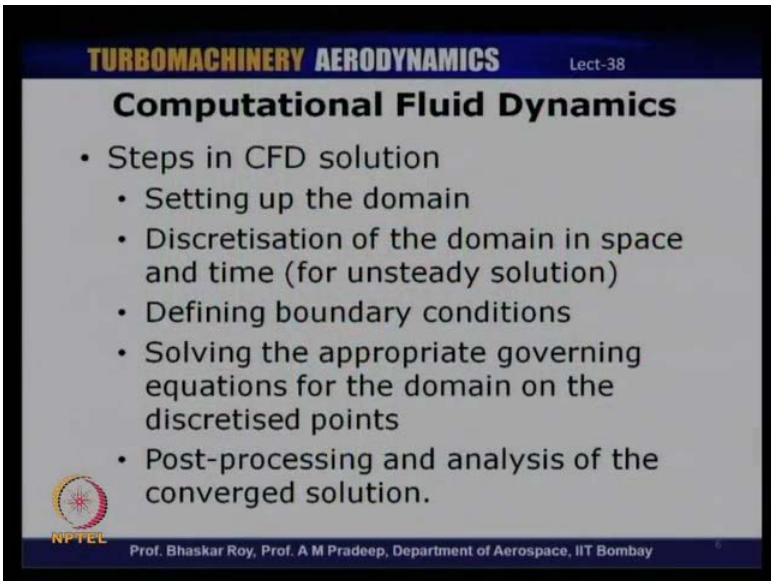
So, these are the governing equations that I am sure you are aware of and depending upon the application some of them may or may not be use. For example, species conservation may not really be used for a compressor flow at least the initial stages of a compressor flow where the temperatures are not very high, one would not expect any combustion or any reaction taking place.

So, in non-reacting flows the use of species conservation does not make any sense. So, one need not use species conservation. So, these are the governing equations that have been solved in CFD analysis and then how do you solve these governing equations. So, there are S series of steps of which are followed in typical CFD analysis and basically the step begins with identification of what you need to simulate. For example, if you need to simulate a compressor flow or flow field around a compressor blade, one needs to define the domains or boundaries of this particular flow field that you are simulating.

So, in a compressor blade let us say we are simulating only one blade of a compressor, then one needs to define what are the bounds or limits around the compressor blade where we need to compute we also need to keep in mind a few things that the domains are not too close to the surface because that would not hel[p]- give us the chance to compute all the flow physics present. It cannot be too far away because that will increase your computing time. So, one needs to have an optimum domain and that is the first step of any simulation to identify the domain or boundaries of the simulation.

The second step is to discretize the domain itself that is you would need to determine the number of points in the domain at which the solutions or or the governing equations are solved. So, all these discretisation is probably the second step after defining the geometry and the domain, one would need to discretize the domain.

(Refer Slide Time: 23:07)



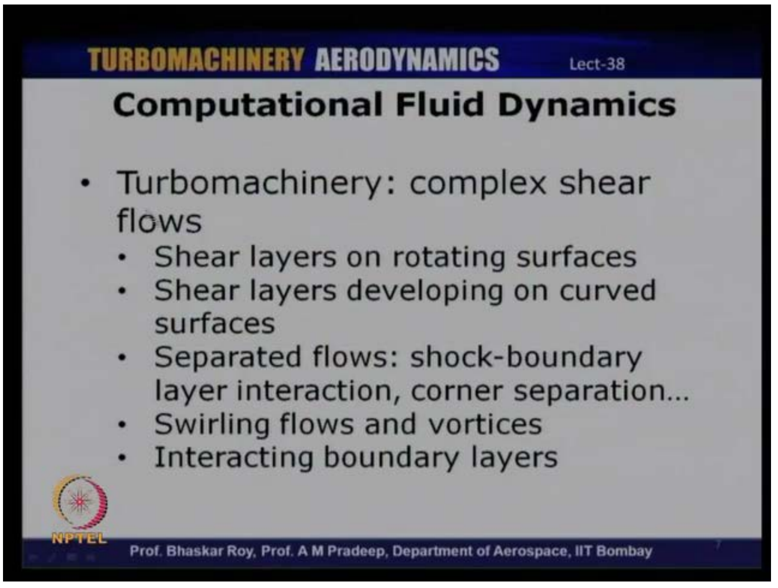
Discretisation would involved basically a machine or gridding the geometry by defining points various points on the geometry and then the those points are where the governing equation would be solved and so, discretisation could be either in the space domain or even in the time domain. One could discretize in one definitely needs to discretize in time well in space, one may also need to discretize in time if you are looking at unsteady solution that is if it is a time marching solution there is also a discretisation in time domain that is after how many times steps that we need to proceed and find out the solutions for the next time step.

Once you discretize the domain, the next step is to define the boundary conditions that is you need to define the conditions at the boundary because that is there the simulations would begin and then and the simulations or the solver would maintain the boundary conditions that one is defining. Subsequently we solve the appropriate governing equations at these discretize points and once the solving is done through a series of iterations and once the iterations have converged as per the convergence criteria that is been specified, one can post process and analyze the converge solution.

So, these are the series of steps that one would follow in a standard CFD solution this is of course, independent of what you are trying to simulate whether it is flow pass cylinder whether it is flow passed an aircraft or flow passed in airfoil or flow passed a compressor blade, the steps that are required to be followed are identical in all these cases. The governing equations are more or less identical, boundary conditions are different, the geometry is of course, different. So, as we are discussing about turbo machinery flows why is it that turbo machinery is very difficult to simulate as I have mentioned in the beginning there are lot of challenges associated with simulating turbo machinery flows.

So, there are a few issues which are associated with turbo machinery flows and probably specific primarily turbo machinery flows which makes simulating turbo machine flow quite complicated and challenging task.

(Refer Slide Time: 25:19)



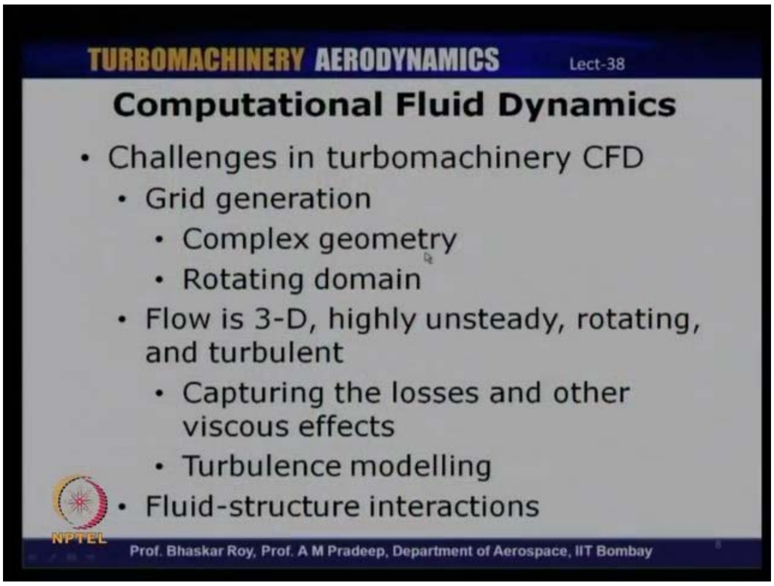
Now turbo machinery involves; obviously, very complex shear flows which involves shear layers on rotating surfaces on the blades. Shear layers on which are developing on the curved surfaces again on the blades, one may have separated flows which could be because of shock and boundary layer interaction or corner separation or during stall. Shock boundary layer interaction being when it is transonic case.

Turbo machinery flows also involves swirling flows, because the flow exiting a set of rotor rotor blades are swirling and there are vortices involved in such flows and you also have interacting boundary layers between the blade surface and the casing or blade surface and the and the hub surface and end wall boundary layers and so on. So, all these put together make turbo machinery flows extremely complicated and it is not possible to simplify this problem one can of course, simplify, but with the loss of accuracy.

For example, if you have to really simulate a turbo machinery flow let us say compressor flow, one cannot simply do a two dimensional Euler analysis and estimate the performance. One could at a as a starting point, but it will no no way give us the exact performance because Euler of course, solution does not give you the losses and if you go for a 2-D NS solution Navier- Stoke solution, one would miss out on a variety of losses like the 3-D losses which are involved. So, one needs to do a 3-D Navier-Stoke solution to be able to estimate the performance in the right way, and therefore to be able to generate a 3-D Navier-Stoke solution one needs to understand the complexities that are involved in this solution. Now there are a lot of challenges as I have mentioned in turbo machinery CFD starting from grid generation.

So, grid generation itself is a challenge because the geometry can be quite complex we have blades which could be twisted and which could have different curvatures has lot of radii at the leading edge and trailing edge and at the junction between the blade and the hub surface all these make the geometry extremely complicated.

(Refer Slide Time: 27:38)

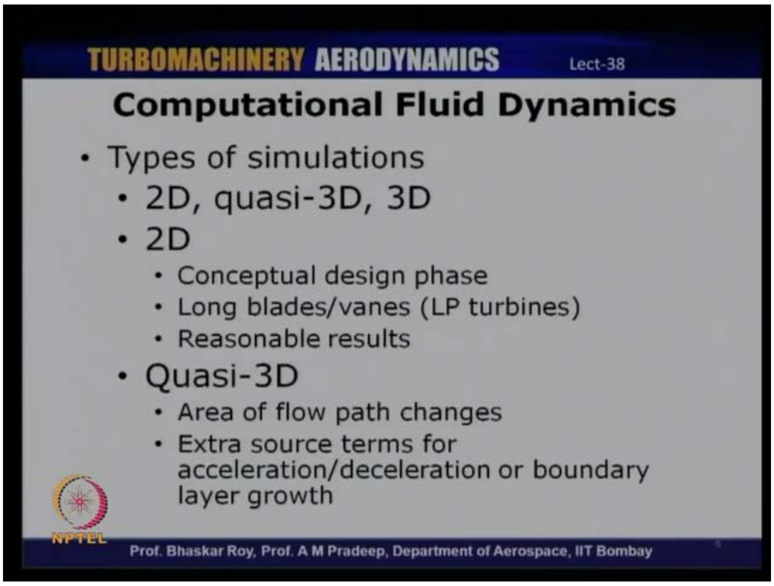


So, grid generation is an extremely involved process in turbomachinery flows plus the fact that we also have a rotating domain. The rotor blades are rotating and the rotating flow goes into the stator and which is stationary. So, how do how do you actually take this into account how do factor the fact that there is a rotating domain which is present in turbo machinery flows.

Besides this the flow in turbo machine is three-dimensional, it is highly unsteady, it is turbulent, extremely complex shear flows as I have mentioned in previous class a previous slide. Capturing all these different effects and and also viscous effects and put together how do you actually simulate all these using let us say Reynolds Averaged Navier-Stoke equation, because there is certain as I mentioned the flow is also turbulent higher turbulent. In fact so, turbulence modeling becomes a very challenging task in most of these turbo machineries CFD's. So, how do how does a designer or an analyst who is trying to work out performance of a turbo machine take these factors into account because the flow is extremely a complex.

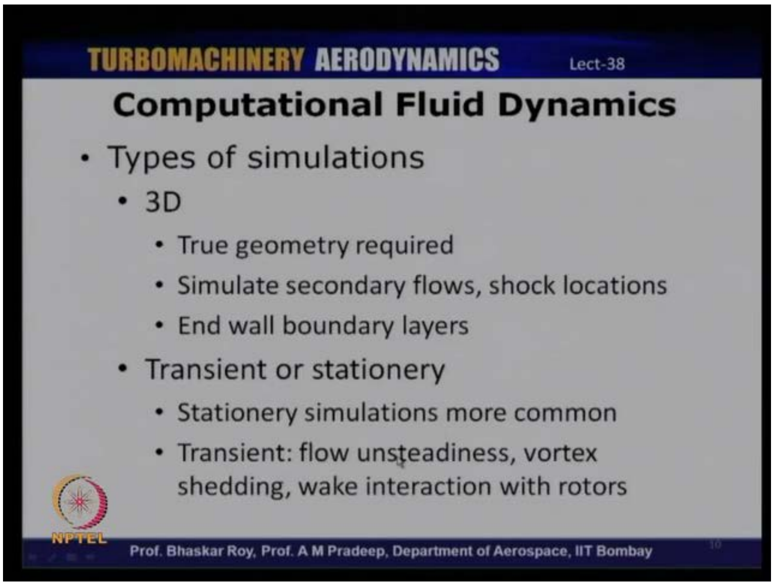
And if you also look at the performance of some of the components like let us say the fan of turbo fan engine. The fan blades tends to deflect and vibrate under the loads the aero dynamic loads and that vibration can also induce an effect on the flow and then there is a back effect on the structure itself. So, there is a very strong fluid-structure coupling and it is also referred to as aero elasticity. So, that is yet another aspect that is quite challenging to simulate and how do you simulate aero elastic effects in some these components like a fan blade. So, fluid-structure interaction is also important which means that CFD also needs to be complimenting other analytical tools like finite element methods which are used for structural analysis.

(Refer Slide Time: 29:54)



So, if you look at turbo machinery CFD we have a few options available, we could go for 2D analysis as I said 2D or you could for a quasi 3D or even a full 3D analysis. A two dimensional analysis could be used on a conceptual design phase where one does not want to spend lot of time on 3 analysis because he would like to first freeze your design and look at whether the design is is conceptually feasible. Well these have been used in in a reasonably reasonably accurate way for let us say long blades where in two dimensional effects can be fairly well simulated. For example, in the LP turbines the last stages of a turbine. If you are not really looking at a very simplistic solution, one can go for a quasi 3D analysis where in the area of the flow path changes which means it is not necessarily 3D, but it is no longer 2D as well and one can add an extra source terms for acceleration or deceleration or the boundary layer growth as a result of area change.

(Refer Slide Time: 31:07)

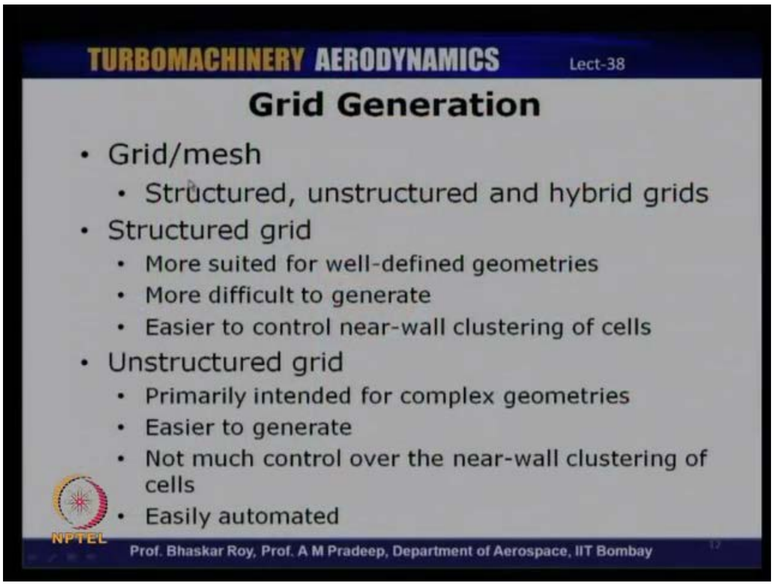


One could actually go for a 3D simulation which is of late being very commonly used where of course, one requires to simulate the true geometry, it can simulate secondary flows shock locations and interactions of end wall boundary layers and so on and so, 3D simulations are usually used towards the end of the design face where we have sort of arrived at reasonably good design geometry. One could also go for a transient or stationary simulations. Stationary simulations are usually or steady simulations usually are more common.

Transient simulations, one can interact one can actually compute the flow unsteadiness vortex shedding interaction of wake with rotors and so on. So, these are simulations which will require a transient CFD run to be carried out. So, these are the different options that a designer has when it comes to trying to simulate a CFD solution. One could also use different types of solvers as I mentioned, one could go for an Euler solver or one could go with a RANS or URANS, LES or probably DNS sometime in the future.

So, what I will take up now are 2 distinct aspects of the solution procedure which are very important aspects of the whole CFD simulation itself. So, we will begin witH-grid generation or mesh generation we will discuss different types of grids or meshes which are used in CFD simulations applied for turbo machinery blades. We will then discuss about boundary conditions in some detail.

(Refer Slide Time: 32:50)

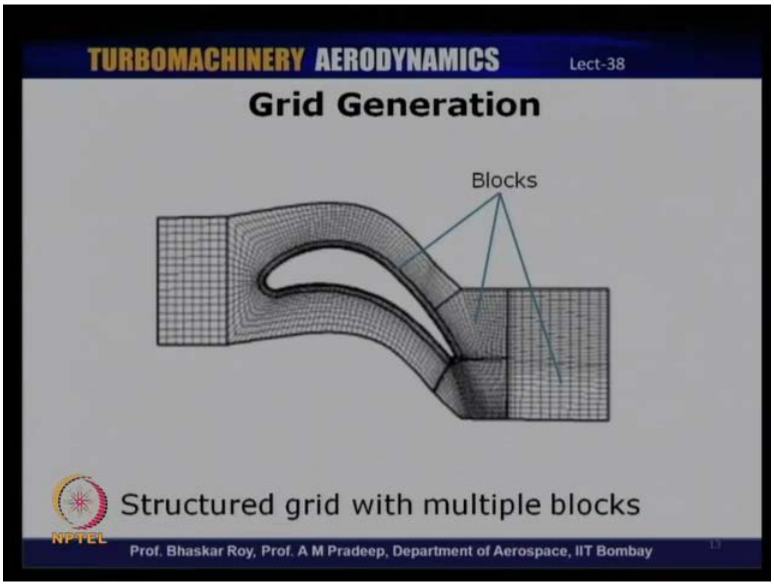


Now grid is the discretisation or grids are used for basically discretizing the domain and there are different types of grids or meshes which are available and which can be used. These can basically be classified as structured grids, unstructured grids and hybrid grids.

For a structure grids are primarily more suited for relatively well-defined geometries, but they are more difficult to generate, but it is possible for us to control the near-wall clustering of the cells very well. That is because, one can change the near-wall number of cells very close to the wall in a much better way using near-wall clustering and it is possible it is gives us more flexibility and control over the size of grids. At the same time, structured grids require less number of well less memory power. The only issue is that it is more difficult to generate and when the geometry is very complex, structured grids may not be very easy to generate.

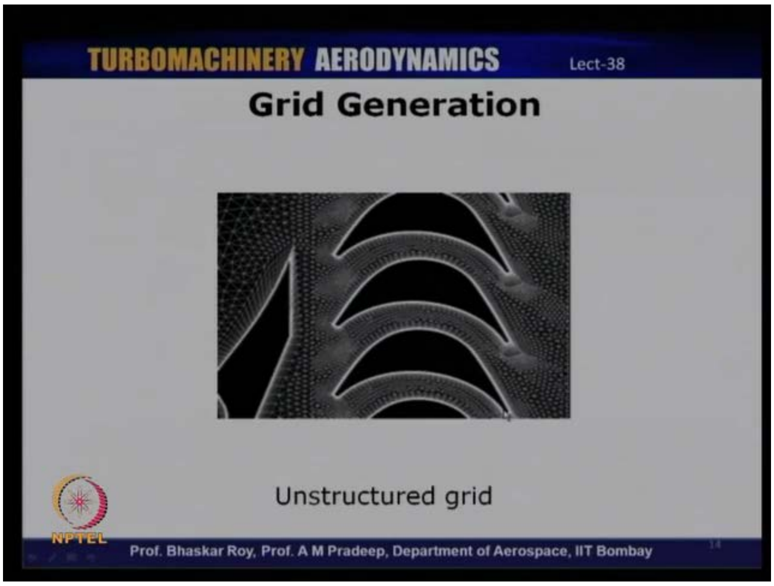
Unstructured girds on the other hand are intended primarily for complex geometries, they are easier to generate it and very easy to automate as well, but the major disadvantage is we do not really have a control over the near-wall clustering, we do have some control, but not as much as control as we have in the case of structured grids. So, a designer would often want to use structured grids for his analysis.

(Refer Slide Time: 34:31)



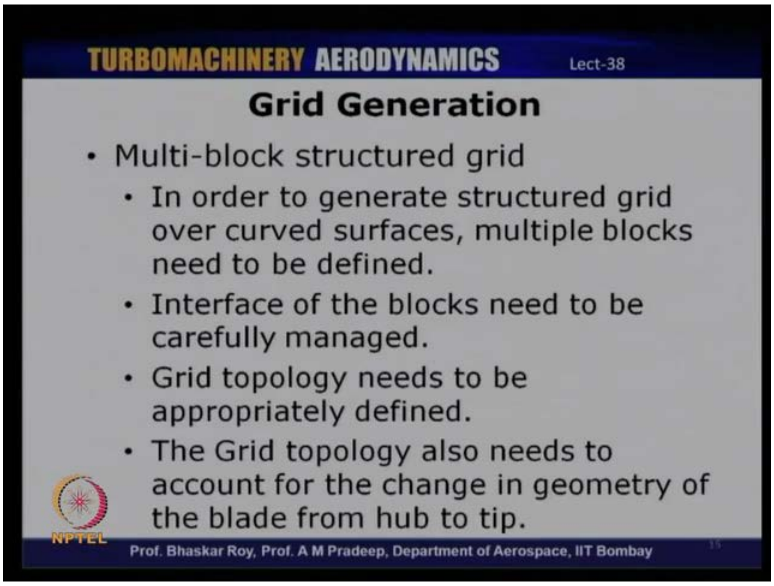
Let us take a look at both of one example of both of these different types of grids. So, this is a typical structured grid and as you can see there is a certain amount of structuring and you can define the grids in certain manner and it is not random. Structured girds are usually used with multiple blocks and I will explain what is blocks and topology little later, but you can see they are distinct where is one block here around the blade, there is another block here, there are blocks here as well. So, this is what is known as a multi block structured grid.

(Refer Slide Time: 35:06)



And this is a typical unstructured gird and you can see that the grids are not in a particular fashion. And they are relatively random and this is a typical example of an unstructured grid. These are unstructured grids you can see that immediately that not very easy to control the clustering around the blades where you would like to simulate the boundary layers as well.

(Refer Slide Time: 35:31)

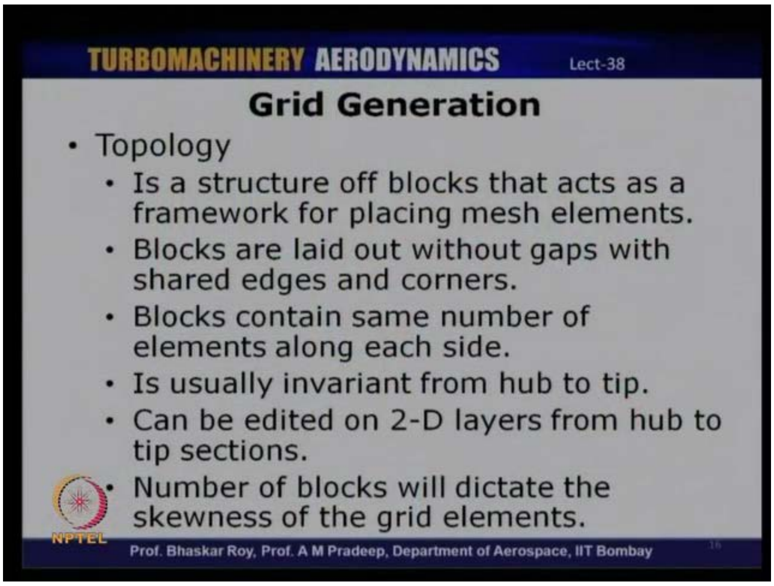


So, let us start with the structured grid multi-block and these are multi-blocks are used if we need to simulate curved surfaces. So, generating a structured grid without multiple on a single block is very difficult it is not really possible without having multiple blocks around the curved surface and the in order to define multiple blocks on the curved surface one needs to define what is known as a grid topology and grid topology also is to it is should ensure that the grid topology will basically remain the same from the hub to tip and so, that grid topology should be able to account for the variations in the blade shape from all the way from hub to tip.

So in a multi block structured grid, one needs to first identify and develop the topology of the grid on the surface and then generate the grids within each of these blocks and at the interface of these topologies or interface of these different blocks the number of cells will have to be the same usually the same and of course, in some solvers they dO-grid generators, they also give us a provision for having variable number of mesh points at the interface. So, then adjacent blocks need not necessarily have the same number of cells, but usually it is a practice used to maintain the same number of cells across the blocks adjacent blocks.

So, when you develop these number of discrete number of blocks of around a surface, one can actually control the number of elements in each of these blocks and that is the main flexibility which a structured grid provides over an unstructured grid.

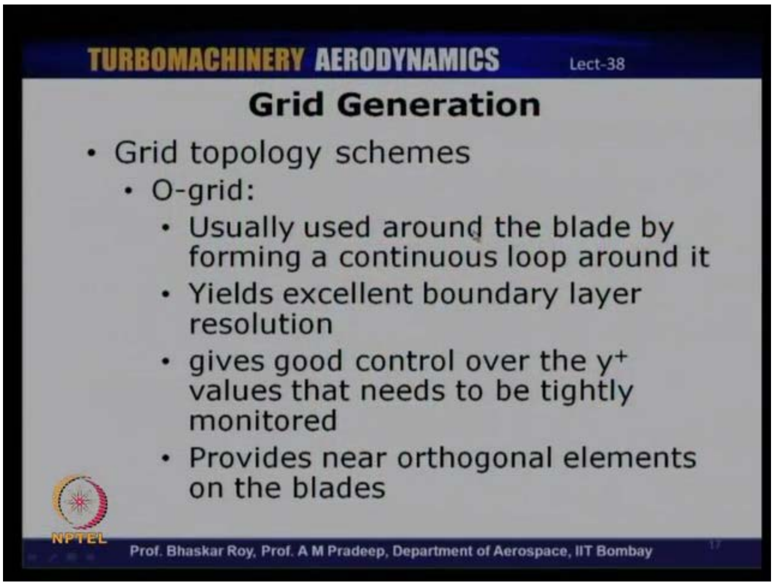
(Refer Slide Time: 37:24)



So, if you look at grid topology it is basically a structure of blocks that axis a frame work for for measuring the mesh elements. So, you have to generate mesh, one needs a structure into which the meshes can be placed or grids can be placed. The blocks are basically laid out without any gaps so; obviously, and shared edges are and corners are possible blocks contain primarily the same number of elements along each side. Topology usually does not change from hub to tip as I mentioned one needs to maintain the same topology from hub to tip and one can edit the topology on 2-D layers from hub to tip which means that if you change it on any of these surface it will basically be applied all over from hub to the tip section.

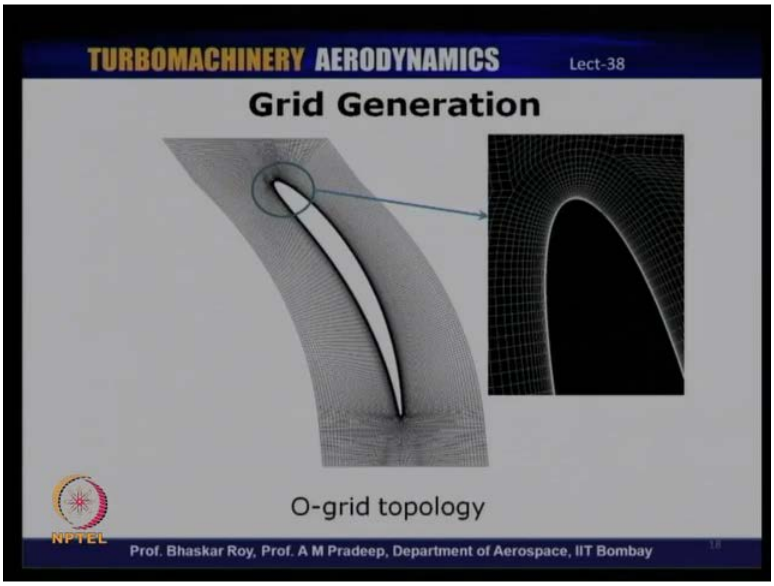
The number of blocks will also determine the skewness of the grid elements that is if you use lesser number of blocks, if you do not have any blocks at all and try to generate a structured grid then what would happen is the regions with where there is a very sharp change in the slope or curvatures the mesh or the grid becomes extremely skewed and such a skewness can lead to lot of issues in convergence of the solution or accuracy of the solution itself and that is why multiple block are anyway required for curved surfaces and use of multiple blocks will give us a lot more flexibility in terms of the mesh or grid management around the surface.

(Refer Slide Time: 38:55)



So, let us look at what are the different types of topology schemes that one can use let us start with the O-grid. O-grid is usually used around the blade by forming a continuous loop around a surface and it gives us very good boundary layer resolution because it is around the blade surface that we would really be like interested in capturing the boundary layer and the viscous loss effects. O-grid gives good control over y plus values that needs to be tightly monitored as you are aware, y plus refers to the nearest element to the surface and to be able to simulate the boundary layers very well one needs to have very low values of y plus. O-grid gives us a very good control over the y plus and O-grid also provides near orthogonal elements of the blades.

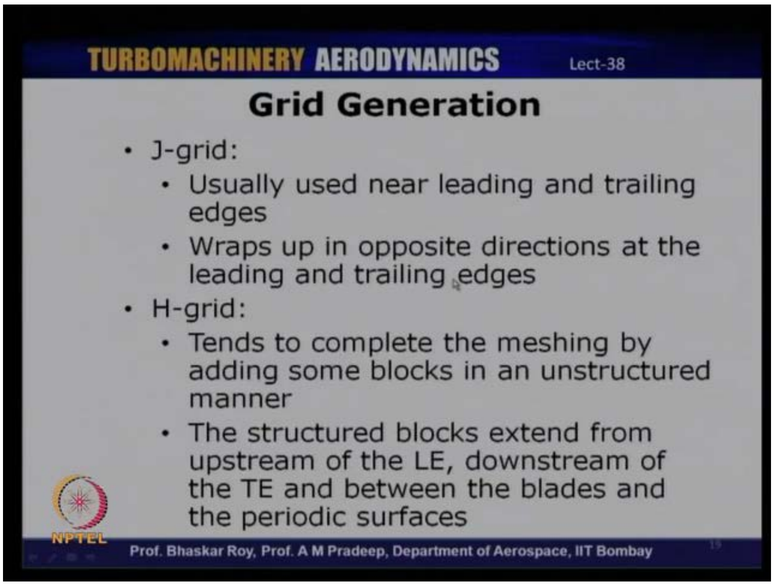
(Refer Slide Time: 39:45)



Now, let us look at a one of the O-grid topology. So, this is an O-grid that you can see around the blade all along the blade there is a set of elements which have been demarcated as you can see the close up of this leading edge of the blade, you can see that this is basically the O-grid which demarcates the grid around the blade from rest of the blades.

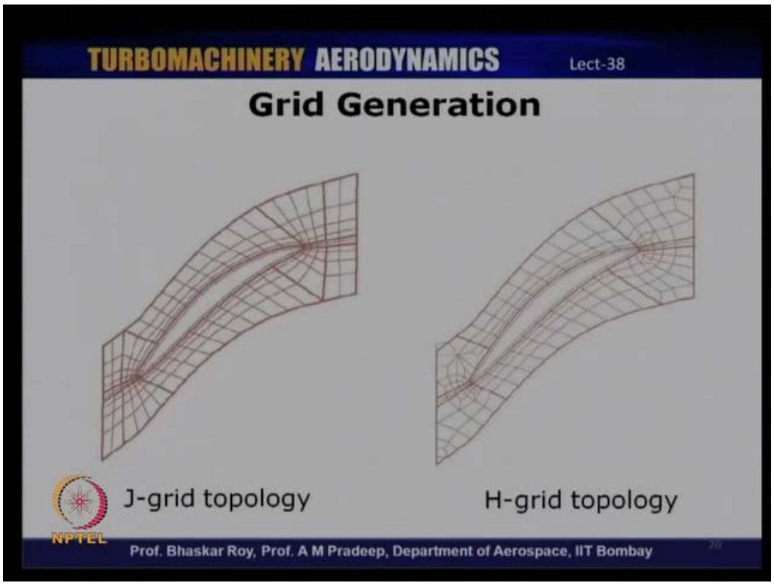
So, there is clear cut demarcation here and that is possible because of the O-grid that is usually developed around the blade surface. So, this is the basic O-grid topology.

(Refer Slide Time: 40:25)



Now there are other forms of topologies which I have shall discuss now there are J-grid, H-grid, C-grid and L-grid. J-grid is usually used near the leading and trailing edges and it usually wraps up in opposite directions at leading and trailing edges. On the other hand, H-grid tends to complete the meshing by adding some blocks in an unstructured manner that is towards the leading or trailing edge one may have some elements of unstructured blocks. The structured blocks usually extend from upstream of the leading edge, downstream of the trailing edge and between blades and periodic surfaces and you may have certain elements of unstructured blocks which the H-grid will develop for us.

(Refer Slide Time: 41:06)

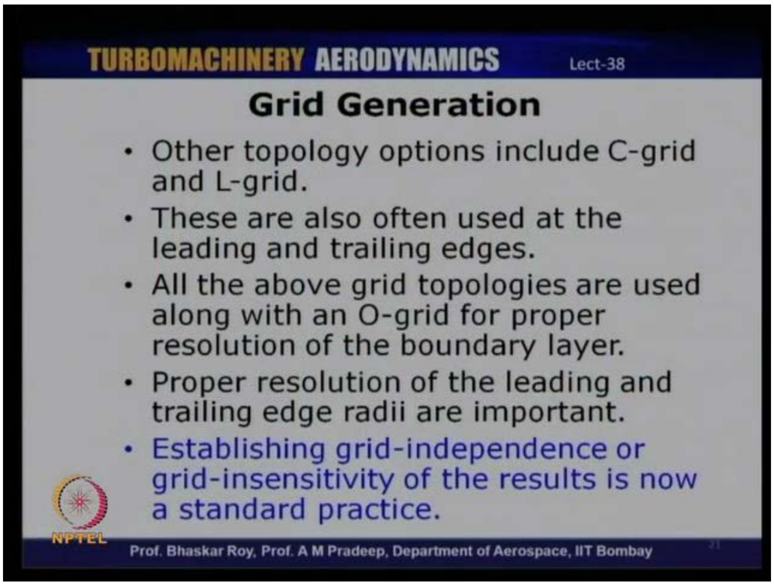


These are two examples of typical J-grid and H-grid topologies and why is it called J-grid because you can see that this resembles a J and that is why there is a set of elements which are resembling the symbol J. So, you can see a J here, this way and the J as I mentioned would be in opposite orientation. So, at the trailing edge it is in this direction. So, this is a typical J-grid topology and on the right hand side you have an H-grid topology.

So, H is resembling the letter H and so, this is basically an H-grid topology and as I mentioned H-grid topology may lead to some amount of unstructured elements towards the leading edge and trailing edges on account of the topology itself. So, if you had introduced yet another block here that would become a J-grid and then you may not require this unstructured block which are present here and all these are in conjunction with an O-grid, you can see the O-grid right around the blade in both J-grid as well as H-grid topology, one can see the O-grid right along the blade surface.

So, most these topologies are usually used in conjunction with an O-grid topology because O-grid is some which gives you control over the boundary layer resolution or the grids around blade surface which is where the boundary layer is one would like to capture that to the best possible extent and so, O-grid gives you the flexibility.

(Refer Slide Time: 42:52)



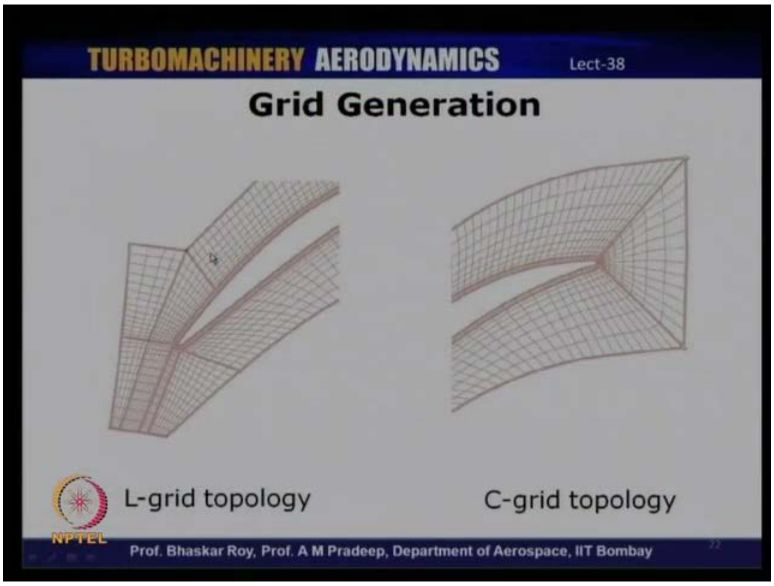
There are other forms of grids which are also sometimes used the C-grid or the L-grid and irrespective of what gird is been used whether it is J-grid, O-gird, C-grid or L-grid.

There are all used in conjunction with an O-grid for good resolution of the boundary layer and it is also necessary that resolve the leading and trailing edges very well, because the leading and trailing edges one has a very sharp in a let us say high speed compressor blades the leading and trailing edges can be relatively quite sharp at a radius radii at leading and trailing edges could be very small resolving these edges are very important because performance of the whole compressor is a function also a function of the leading and trailing edge radii.

Now, irrespective of what kind of grid we use or which application is to be used for one needs to establish that the grids or the solutions that you get from the computations are independent of the grid sizes that is if you use 1 million cells or 10 million cells is there is a difference in solutions then which one do you believe one would want to believe which has more number of cells. So, it is a common practice now standard practice now to establish the grid-insensitivity or grid-independence of any simulation.

So, one needs to demonstrate that as you change the number of grids from certain number to the double that number or half the number, the number of the depending upon how many numbers you have changed the solution should not be dependent on the number of grids that you are using, and therefore one would of course, like to optimize between number of cells used and the best possible solution and that is where one would want to do a grid sensitivity analysis and establish that the number of grids you are using for analysis is optimum and the solution for that particular number of grids and anything above that would remain the same. So, one needs to establish grid-independence irrespective of what kind of topology are you using.

(Refer Slide Time: 44:59)

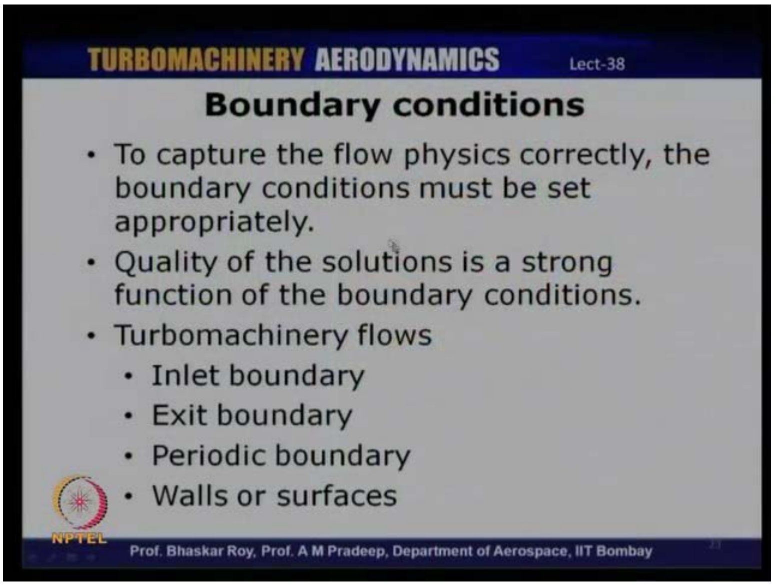


And let us take a look at 2 other grid topologies the L-grid and C-grid. This is the L-grid topology very similar to a J-grid, but just that in a J-grid one had a section here and this is basically the L-grid topology that you can see and this is a typical C-grid topology you can see that unlike the multi-block here multiple number of blocks here you have only 2 blocks which have been provided and this is the typical C shape that you get and that is why it is called a C-grid topology.

So, which topology to use basically depends upon the application and there are certain standard thumb rules which are available which can help an analyst making a decision on which kind of topology that he needs to use for his application. So, I did not understood some elements of grid and grid generation. Let us look at the other aspect of starting the simulation that is basically the boundary conditions. Boundary conditions are; obviously, extremely important for a particular solution and its accuracy because the solution primarily depends upon what kind of boundary conditions you are setting.

So, depending upon a different set of boundary conditions the solutions can entirely be different. For example, if you look at a compressor operation, if you set the design boundary conditions right one can get the design operating conditions or simulations for that operating conditions, but if you get the design operating or the boundary conditions which lead towards compressors. So, as then the solutions are entirely different from what it should have been, and therefore to get the correct flow physics and to be able to make sense out of the simulations one needs to ensure that the boundary conditions are set correctly.

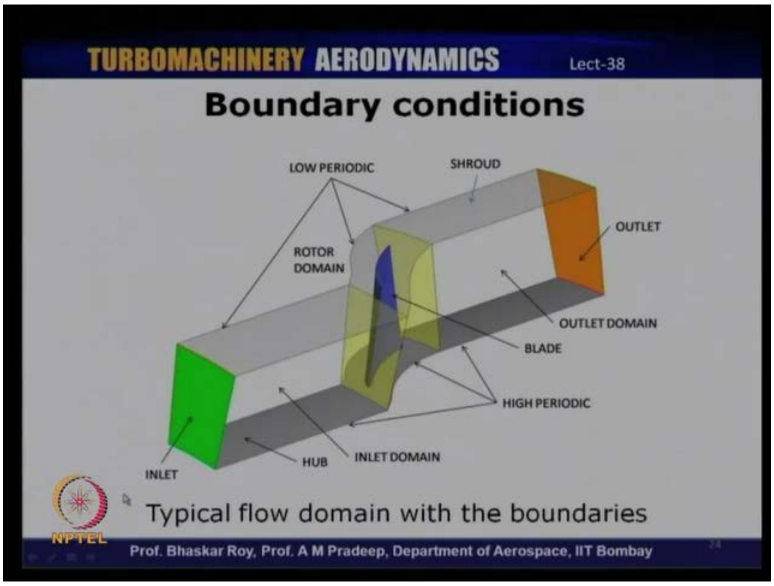
(Refer Slide Time: 46:49)



So, basically boundary conditions are essential in capturing the flow physics correctly and appropriately and; obviously, the quality of the solution that you get is a very strong function of the boundary condition itself.

In a turbo machinery flow, there are of the 4 distinct types of boundaries that one would encounter. The inlet boundary from which the flow begins or initiates, the exit boundary that is at the outlet of the flow domain, periodic boundary on 2 sides of the domain and walls or surfaces which could be the blades could be the hub or the casing.

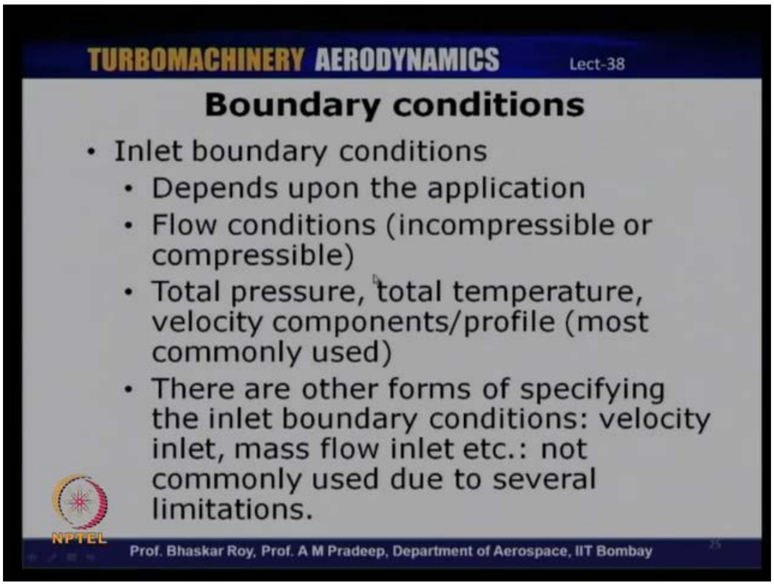
(Refer Slide Time: 47:27)



So, this is the typical flow domain with the boundaries I have shown. This is compressor blade here we have a low speed compressor blade the flow enters through the blade domain and here we have seen relating only one blade and since all the blades are identical we will have to set what are known as the periodic boundary conditions.

So, this is the inlet domain that you can see here as the inlet, this is the hub surface of the of the blade and this is the shroud of the tip. The flow enters through the domain here and then it exits through the outlet which is shown here in an orange and this is known as the outlet domain and on the sides of the blade we have what is known as periodic boundary conditions which means that whatever happens here will be a reflection of what should he happening even if they work in number of other blades it around it and the periodic boundary walls will depend upon the number of blades basically depending upon the solidity itself, if you have more number of blades then these boundaries would come closure. In this particular domain, this is the domain which rotates and these are the 2 domain that is the inlet and outlet domains are the one which are stationary. The rotor domain is the one which rotates.

(Refer Slide time: 48:45)

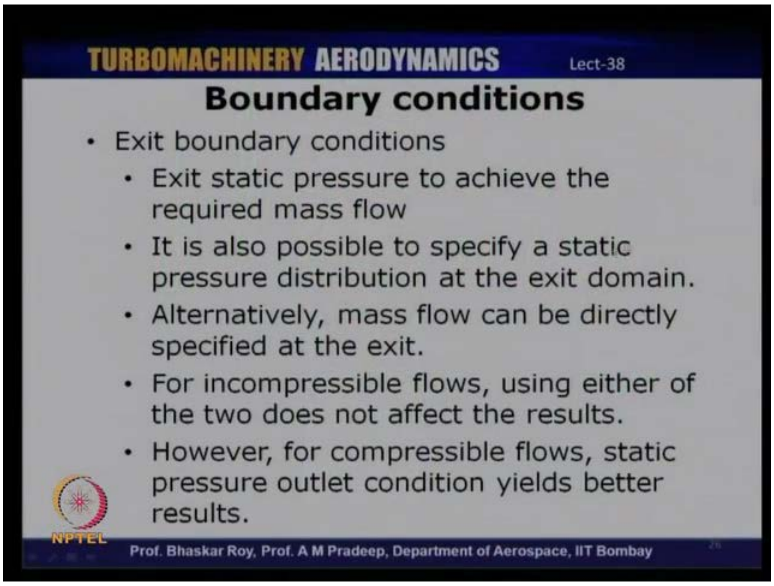


So, if you have to set inlet boundary conditions it; obviously, depend upon the application itself which depends upon the flow conditions whether it is incompressible or compressible, the most commonly used boundary condition for the inlet is this a set of predefined total pressure at the inlet, total temperature and the velocity components or profile at the inlet.

So, this is most commonly used form of inlet boundary conditions which are applied in a standard CFD solution for compressor or turbine flows. There are other forms of specifying boundary conditions it could be velocity inlet which if it is very low speed in compressible flow, one might even want to specify just velocity inlets or a mass flow inlet and there are these are not generally used because there are lot of limitations with specifying this as compared to the total pressure, total temperature velocity components because that is specifying velocity or mass flow rate alone is too simplistic boundary condition to be applied which probably is or simple two-dimensional analysis if you one would carrying out that, but not generally used for series serious high and CFD dissimulations. Similarly at the exit, one needs to specify a set of boundary conditions.

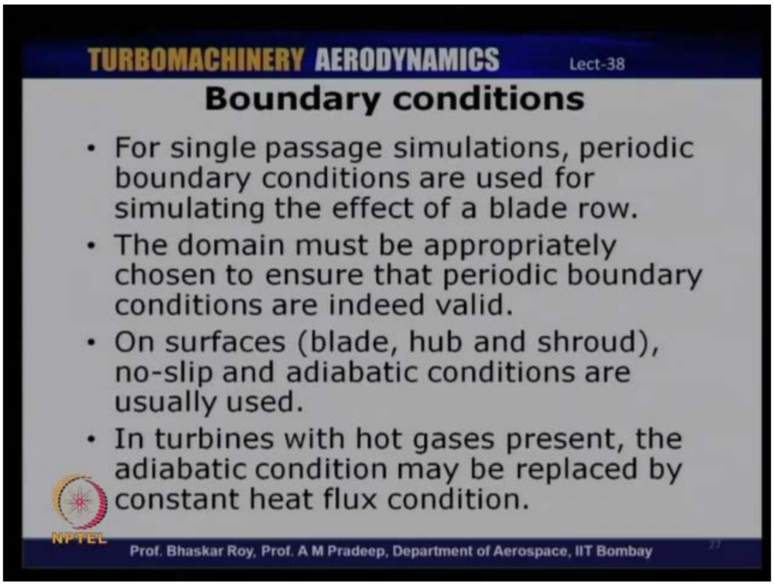
Now here again there are different ways of specifying boundary conditions, most commonly used are either static pressure which can for a given inlet pressure give you a certain amount of mass flow rate. The other boundary condition that one could use is mass flow rate itself and for low speed incompressible flows whether you specify a static pressure or mass flow rate the results are not going to be highly different and they are not very sensitive to this boundary condition, but for high speed compressible flows specifying static pressure at the exit is the standard practice and one would and also it this is a common practice even now used for low speeds simulations it has been sort of establish that a specifying static pressure is probably a better way of defining mass flow rate at the exit of the domain. So, one could specify either static pressure at the exit or just the mass flow rate itself

(Refer Slide Time: 51:06)



and but I say I mentioned for low speed incompressible flow it should not really matter whether you are specifying static pressure or mass flow rate and it is also possible that one can specify a static pressure distribution at the exit rather than just an average static pressure, one may also want to be a little more accurate, one can simulate a static pressure distribution as well at the exit.

(Refer Slide Time: 51:35)



Now, for single passage simulation like the one I have just showed periodic boundaries are used for simulating the effect of a blade row and it is necessary that one appropriately chooses this domain, so that periodic or periodicity is indeed valid. And on the surfaces like the blade or the hub or the shroud it the standard boundary conditions like no-slip and adiabatic conditions are used.

But if you look at a turbine flow which has hot gases present if you are also simulating the hot gases effect in a turbine flow instead of adiabatic condition one may replace that with a constant heat flux condition . So, these are the two periodic conditions that I mentioned and selecting this is very important because this basically represents the solidity of the compressor itself and it simulates the curvature on either sides of the blades. So, these are the different types of boundary conditions that one can simulate, one can actually one needs to specify at the inlet exit and the side walls and surfaces to be able to simulate the performance and flow conditions and flow physics run correctly.

So, let me now quickly recap our discussion in today’s lecture and we had a very quick discussion on CFD in general, this is as I mentioned in the beginning this assuming that you have some knowledge of CFD already and so, we are trying to look at CFD in general, but we are trying to understand aspects of CFD in in the context of turbo machine flows and in that context I mentioned about two distinct aspects today that is the challenges involved in grid generation to which I also explained the different types of grids that are possible for a turbomachinery blade simulation and the key challenges involved in grid generation. Subsequently we also discussed about boundary conditions and the right way of setting boundary condition for turbo machinery flow simulations.

So, we will continue discussion on some of these aspects in next the couple of lectures as well in the next lecture I intend to introduce a few other challenges which are involved in turbo machinery flow simulation to do with basically the turbulence modeling and associated problems are shown with turbulence modeling and why is it necessary that we use the right model for a set of problems that we are trying to analyze. So, we will take up a few of these aspects in the next two lectures that we would going to have on CFD as applied for turbo machinery flows and next lecture we will take up some more challenges which are specific for turbo machinery CFD. So, we will take up some of these topics for discussion in the next lecture which would be lecture number 39.