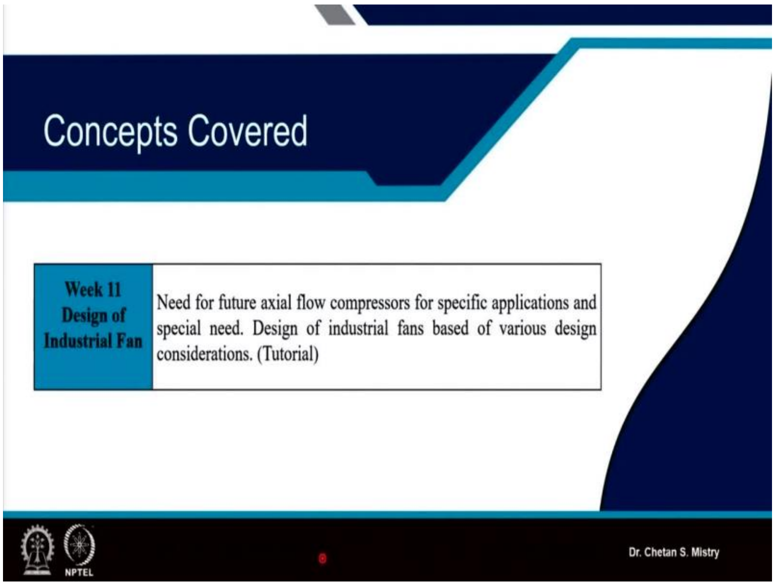
Aerodynamic Design of Axial Flow Compressors & Fans
Professor Chetankumar Sureshbhai Mistry
Department of Aerospace Engineering
Indian Institute of Technology Kharagpur
Lecture 65
Design of Industrial Fan (Contd.)
CFD Application to Design and Performance Assessment

Hello, and welcome to week 12. This week 12 is dedicated to CFD applications for design and performance assessment. We know we are more excited to learn CFD and that to when we are talking about the CFD applications for turbo machinery or CFD application for special requirement, it is of great interest to all of us.

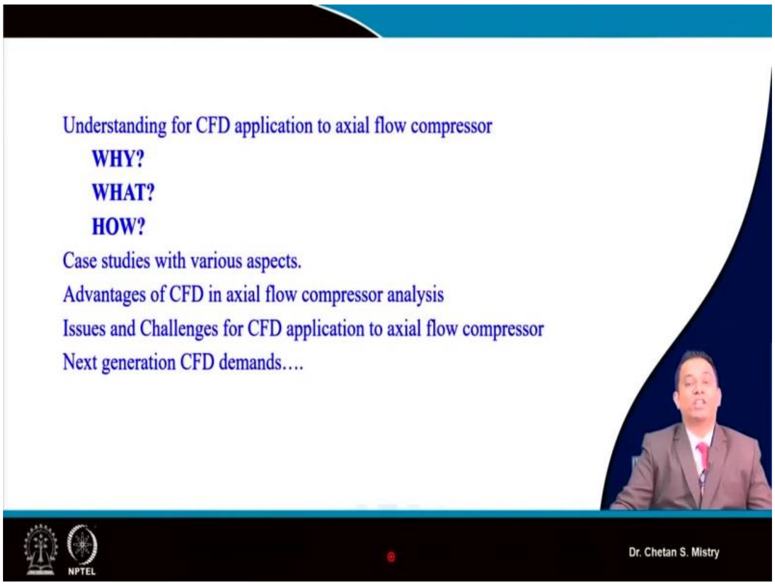
(Refer Slide Time: 00:55)



What all we have discussed in our last week that's what is talking about, say need for future compressors for special application and special need. We have discussed different approaches in order to improve the performance, in order to improve the fuel economy for the engine, in order to address the issues related to achieving high pressure rise, all those aspects we have discussed with.

We also were discussing about the recent trends for the development of axial flow compressor. Then we started discussing about the application of industrial fans. We have discussed about various applications, then we have discussed about the special application of axial flow fans, that's what was followed by say the design of axial flow fan for wind tunnel application. We have also discussed the design of ventilation fan, and that's what was giving the brief idea about design of industrial fans.

(Refer Slide Time: 02:00)



So, in this week, we will be discussing about, say understanding for CFD application to axial flow compressor. So, always the question that's what will be starting in our mind, that's what is why, what and how. Very first question that's what will be coming is say, why we want to do CFD? Now, be careful, people used to make a joke saying colorful display, Yes! If you are not having detailed understanding what you are doing, and then that's what is colorful display only.

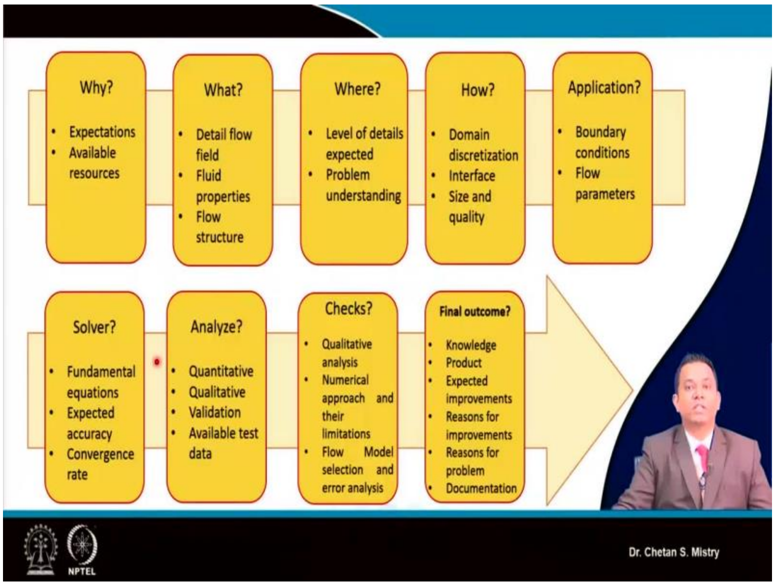
So, the thing is the question of why, what and how that need to be understood by all of us, okay. Then we will be discussing about say case studies that's what will covering different aspects which I explored personally, as well as my group, they are working at IIT Kharagpur. We will be taking some case studies, that's what will be explored by different researchers throughout the world.

We will be discussing what all are the advantages of CFD, specially for axial flow compressors. We will be discussing about say issues and challenges of CFD application to axial flow compressors. And what all are the next generation CFD demands? So, this is what is not the class, that's what we will be starting from fundamental in order to discuss the CFD. That's what is all the way different domain and that's what is required all other kinds of understanding.

So, we will not be discussing about the CFD analysis in sense of different numerical techniques, different equations, no. We will be discussing about what all we are looking for, when we have done our preliminary design. So, if you look at all 11 weeks, we have discussed whole lot of things in sense of aerodynamic design of axial flow compressor as well as fans.

Now, the next step is to go for CFD analysis in order to achieve what we are looking for in terms of performance assessment. And, we need to verify our design what design we have made is it okay to move forward with. So, with this, let us start with.

(Refer Slide Time: 04:26)



So, here in this case, these are the steps what we need to understand when we are discussing about the CFD application. Let it be discussed in sense of general sense of application of CFD as well as CFD applications to turbomachinery, CFD application to say axial flow compressor.

Very first question that will come into mind, why? Why do you want to do CFD analysis? So, the thing is, we are looking for certain kind of say performance expectations, what we have designed? Or maybe already available system, we need to verify is it giving what we are looking for? So, if you recall, we were discussing about the sizing problem, we were discussing about the rating problem, for axial flow compressor. So that’s what is coming here.

Then the next thing, that’s what will be coming in mind is what all are the available resources with us, when we are talking about the design of axial flow compressor, or say making of axial flow compressor, it is but obvious we need to go with the testing, we need to go with the experimentation that may or may not be possible for many universities, maybe for many industries, okay. Under that condition, this CFD, that’s what will be serving our purpose for initial first cut design.

Next question, that’s what is say, what all we want to analyze what all we are looking for? Now, when I say what that means, I need to understand the detail flow field study, what all are the changes of fluid properties, what will be my flow structure; and based on that, we will be analyzing the loss models. Once we are able to analyze these losses that’s what is happening in axial flow compressor, that’s what will be helping us in order to assess the efficiency.

Now, the question is, we are looking for efficiency to be higher, that means, we are looking for these losses need to be reduced. So, until and unless we are having detailed understanding of flow field, it is difficult to analyze those losses or to improve the efficiency. Now, the question is where? So, you know, level of details what we are expecting, and the problem understanding. So, the thing is we need to realize what is the application of what we are designing. Until and unless we are having the background of that, what all analysis or what all design we are doing it is of no use.

So, we need to be very careful, what is the application, where you are applying this logic? Now, after knowing all these aspects, now, the question is how to do this analysis? So, what we will be doing is, we will be making our computational domain. Now, this computational domain, that computational domain, we need to discretize or we need to divide into small - small volumes. And, that’s what is called a domain discretization.

Then we need to understand say we are having rotor and stator that’s what is need to be incorporated with say interface. So, we need to have the details of interface, we are looking for say size and quality of mesh.

Now, what we are discussing at this moment, size and quality of mesh, you can imagine we have discussed the design of say wind tunnel fan that’s what is having diameter of 4 meter; we have discussed about ventilation fan, where we were having a diameter of 0.28 meter. Now, all this, that’s what will be asking for discretization of the domain. Now, that’s what is coming into the picture, it need to be realized like how many number of nodes, how many elements we will be selecting in order to do simulation. So, this is what is very important. So, we need to know how to do that part.

Now next, that’s what is where we are applying this, or what is the use of what we are designing? That’s what we will be deciding with what all will be the boundary conditions. So, suppose say we are designing say fan for Aero Engines, we can say we have an entry condition as atmosphere, my exit condition that is also maybe with somewhat high pressure rise. Suppose say we are doing our design for LP compressor, we are doing our design for say HP compressor.

So, we must know what is my entry condition and what will be my exit condition. Until and unless we are having this detail design or say detailed information, it is very difficult to analyze that part. So, we need to have the idea of that. What all flow parameters we are looking for that is also equally important.

Now, after doing this understanding, we have our discretized domain, we are having our boundary conditions. Now, we are looking for the solvers. So, solvers basically they will be solving our fundamental equations, we can say conservation of mass, conservation of momentum, conservation of energy, maybe we will be having say perfect gas equation, all those things that need to be solved.

Once we are solving that based on that we are doing calculation of different parameters, or different properties, maybe in terms of pressure, in terms of temperature, change of density, all those parameters, change of velocity, those all parameters, that’s what will be coming into the picture.

Now, for solver also, we need to have idea about what expected accuracy we are looking for. And, that’s what will be deciding what kind of solver we are selecting. Then we need to have idea about the convergence rate. Now, the available computational resources, that’s what will be putting constraint here.

So, we need to be limited, suppose say if we are limited with the computational resources, under that condition, we will be going with the solver with some low configuration. If we are having say very good computational facility, we will be going with say solvers, that’s what is of high configuration. So, selection of the solver also is very important.

Now, after doing all this simulation or analysis, we need to analyze our result. What we say in sense of analyzing? Maybe quantitative analysis, qualitative analysis, we need to go with the validation, suppose say experimental results are available with us that’s what will be the best reference in order to assess what simulations we have done.

So, until and unless we are having this data for validation, this computational analysis that’s what will become or say colorful display only, okay! Now, the thing is, we are looking for a variable test data. Now, this test data, we know the experimental part, that’s what also will be having certain limitations. So, we cannot go with detailed flow field study until, unless we are having expensive, say, maybe optical tools.

So, with using, say, probes…pressure probes, or maybe by using say hot wire probes, we can do our measurement, that's what will be giving quantitative readings; but in detail flow field, that's what will be difficult and that is where the CFD is coming into the picture.

Next, that's what is we will be doing the checks by qualitative checking, that's what we will be doing with the numerical approach and what limitations that's what is having the flow model selection and error analysis. So, once we are having our experimental results available with us, we will be validating that with what simulations we have done. And, that's what will give the error analysis or that's what will be giving the confidence in sense of what we have done as CFD simulation.

Now, once all these details, that's what is available, very important part that's what is say final outcome. So, what we will be learning, how my flow that's what will behaving in the flow domain, that's what is giving my say, knowledge Base database, it will be giving us say final product, expected improvements what we are looking for; suppose say already designed compressor or fan, that's what we are having, maybe we will modify certain geometries, we will be modifying certain parameters, that's what will be giving my expected performance improvement.

Say recent trend, that's what is to enhance, say, the operating range. So, what all modification we can do such that without compromising in terms of efficiency, we can move with increasing the operating range, that's what it is very challenging. That's what is possible by exploring this computational fluid dynamics analysis.

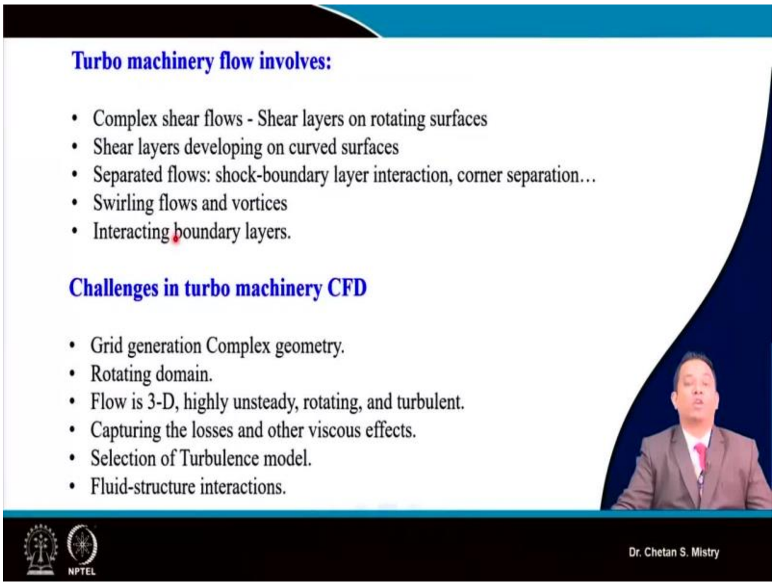
Now, the thing is, we also must understand what all are the reasons for the improvement what we are getting by this analysis, okay. And, that's what will be helping us for the future designs. Now, documentation of what all we have done, that's what is very important. So, in overall, if you look at this computational fluid dynamic analysis for axial flow compressor, or the fan, that's what is demanding for all these aspects. And we must realize why we are doing this computational fluid dynamics analysis.

Many times, people used to say this CFD is a design tool. It is a misconception. What all we have done in last 11 weeks, that's what is your design, that's what is your aerodynamic design. That's what is your preliminary design, based on that you are making your blade geometry. Now, after making that blade geometry, you are analyzing your blades. And once you are analyzing your blade, that's what will be giving what we are looking for in terms of performance of those blades or those rotors and stator combination or that stage or maybe multi stage configuration.

So, do not be in a misconception saying like CFD is a design tool, rather we can say CFD is design improvement tool, okay. There are many commercial companies, they are giving say you can do design quickly by putting the numbers, that's what will be generating the geometry, after generating the geometry, you can do your CFD analysis, you can check with the performance assessment, it is not that easy when we are talking about the application of special need, be careful about all those aspects, okay.

So, we need to do our own design, we need to analyze our own results, and then after you come up with the solution, okay. So, this is what is all about the CFD, how it is working, and how we need to think of.

(Refer Slide Time: 16:04)



Now, let us move. For say turbomachinery application, when we are talking of, it says we will be having complex shear flows. So, we will be having shear layers in rotating surfaces, shear layers that's what will be developed on the curved surface, we will be having separated flow, we will be having shock boundary layer interaction, we will be having corner separation. So, all these what we are talking of that's what will lead to have the secondary flow that's what will be generating within the flow passage.

When we say this is what is generated, that's what will lead to increase the losses, what losses we say as aerodynamic losses. And, basically those aerodynamic losses, that's what is deteriorating your performance as well as efficiency. We are having our flow to be swirling flow or it will be having vortices, okay, the interaction with the boundary layers. So, all those aspects, that's what will be coming into the picture, when we say aerodynamics of axial flow compressor.

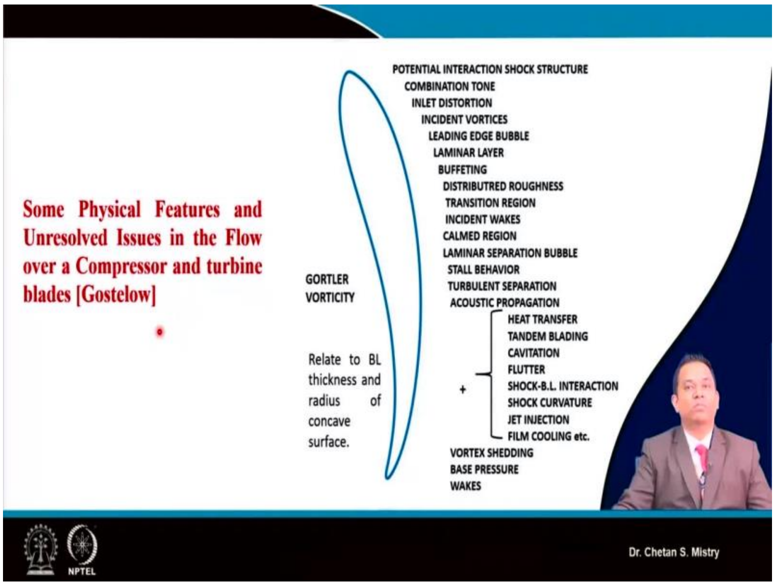
Now, when we are talking about application of the CFD, there are certain challenges. And, those challenges we can say, that is very first thing that's what is for the generation of grids that means in order to discretize your fluid domain. We know our blade shape, what we will be getting using different approaches, say different whirl distribution, that's what will be giving the three-dimensional shape to my blade. And for that, we need to do the discretization, we need to do the meshing, that's what is very challenging.

Next, that's what is we are having rotating domain. My flow will be highly three-dimensional, unsteady, rotating, and it will be having its own turbulence intensity. The capturing of these losses under viscous effect, that's what is challenging. The selection of turbulence model, we can understand at the initial stage suppose if you are looking for the flow over airfoil, initial stage, we are having a flow to be laminar flow, then we will be having transition and later part that's what is turbulent flow, fully turbulent flow.

So, we can understand, we need to capture all this flow physics that's what is very challenging. And along with that, my blade is rotating blade. So, you are having rotating flow with this flow over airfoil kind of configuration, that is the reason why challenge that's what will be increasing. After that, we need to realize the fluid structure interference, or fluid structure interaction, that's what will be giving us say, aerodynamic loading on the blade and that's what is coming in sense of aero elasticity, okay.

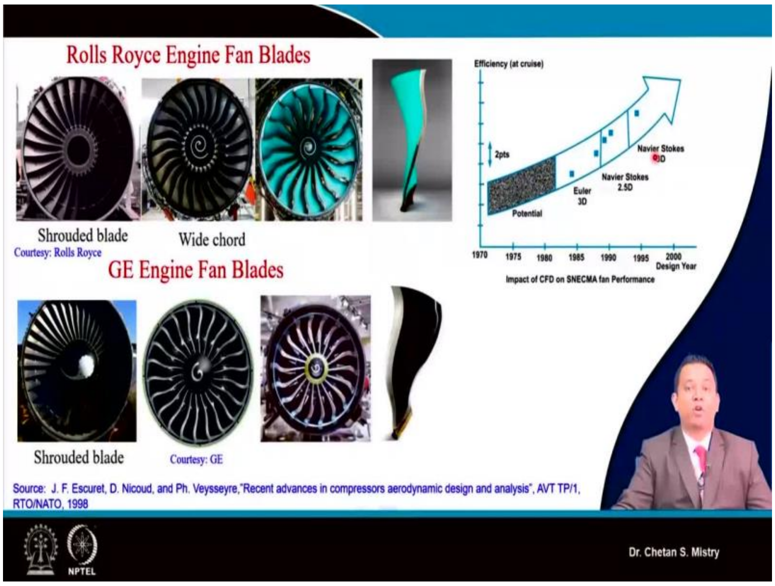
So, all this, that's what is coming into the challenge. So, the analysis of say, turbo machines or CFD analysis of axial flow compressor that's what is very challenging aspects. So initially, it says, like the CFD tool, that's what was initially been developed for analyzing turbo machinery. And then people, they have found their application for different applications for aerospace. Later on, for analyzing different process industries. But very first CFD analysis, that's what was been developed in order to analyze the flow through this turbo machines.

(Refer Slide Time: 19:46)



Now, we know our axial flow compressor, our axial flow fan, that's what is having heart that's what is airfoil; and, now in airfoil as we have discussed earlier, we will be having all this kind of difficulties, that's what is happening on my blade suction surface. Same way, on pressure surface also we will be having growth of boundary layer because of the curvature. Now, this airfoil, this analysis, that's what is very challenging, we must realize that part.

(Refer Slide Time: 20:25)



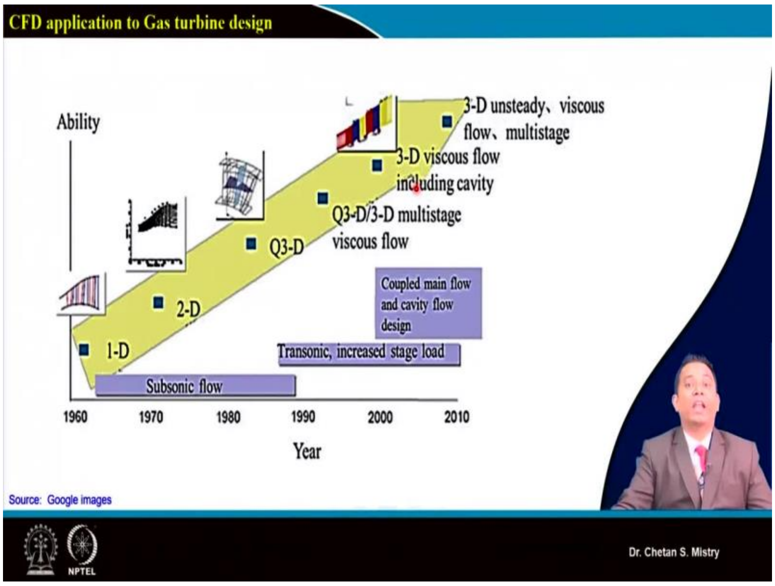
Now, in order to have say use of CFD, so this is what is one of the plot that's what was given by SNECMA for say their fan development. So, if you look at, initially they have gone with the potential flow theory, later on, they have used Euler's 3D model, Navier Stokes 2.5

dimensional model, Navier Stokes 3D model and based on that over the year, they are able to improve the efficiency near the cruise condition, okay.

Now, here if you look at, we have discussed earlier also, these are the rotor blades for fan, that's what has been developed by Rolls Royce as well as by GE, thanks to CFD in order to come up with the latest blade.

So, here if you are looking at, these blades are say not the shrouded blade, here they are having all three-dimensional shape. And, this blade what we are talking of that may be having height in the range of say 5 to 6 feet, and, that's what you need to do design with. So, this computational tool, that's what has helped a lot in order to design such kind of high bypass ratio fans.

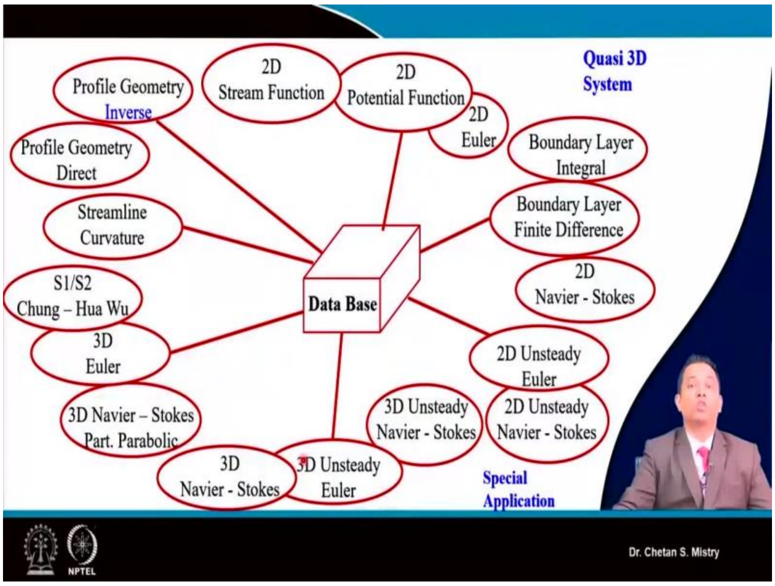
(Refer Slide Time: 21:44)



Now, this is what is a growth of application of CFD over the years. Initially people, they were doing one-dimensional analysis, later on they have done say two-dimensional analysis, Quasi 3D analysis; then later on, as per the availability of computational power and detail flow field understanding, people these days, they are talking about 3D and say unsteady flow field analysis.

So, initially we know our flow, that's what was say subsonic, later on that goes to say transonic and, you know, now we are having say different kinds of configuration, that's what we have defined as say in some region we will be having flow to be subsonic, in some region we will be having a flow to be supersonic and this is what needs to be used in order to finalize our design. Let me put it is not used for design, this is what is to finalize the dimensions and design of say axial flow compressor as well as say axial flow fan.

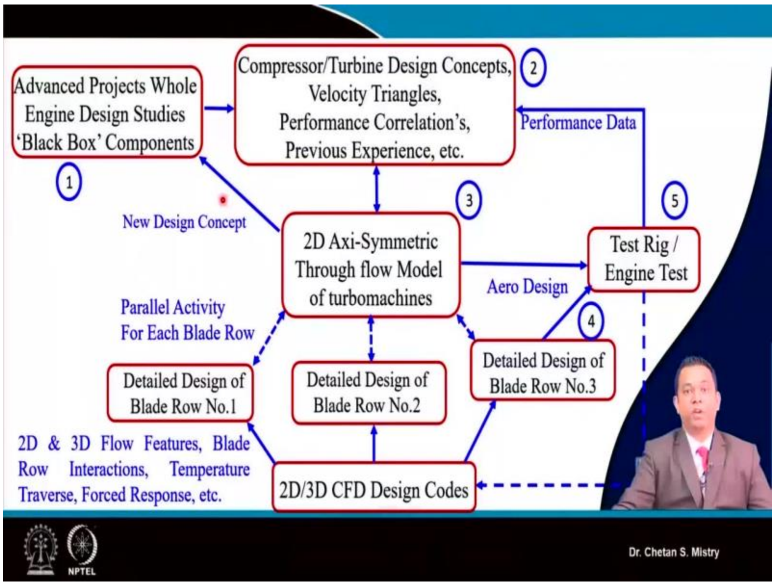
(Refer Slide Time: 22:51)



Now, this is what is whole database that’s what is available with different engine design manufacturing companies. So, different design companies, they are having all these database, that’s what is available with them. And, they are adopting those concepts in order to finalize their design, they are having their in-house code and those codes they have been developed over the year with maturity and understanding with experience.

So, here if you look at, this is what is talking about Quasi 3D; here, we are having say special applications of tool and we are at this moment we are moving in sense of 3D Navier stokes, unsteady simulation, all those analysis that’s what is going on in order to do finalize of the design.

(Refer Slide Time: 23:41)



Now, this is what is a whole cycle. Now, as we discussed, people they are going more aggressive in sense of expected pressure rise, we are looking for say axial flow compressors and fan that need to generate per stage pressure rise to be very high under that configuration, this is what will be acting like a black box.

Now, what we need to do is we will be doing say initial design or say, you know, preliminary design, what all we have discussed up till now. People, those who are working in this aerodynamic design aspect or mechanical design aspect, they are having their previous experience, they are having whole database that's what is available with them. With that they will be doing say 2D axisymmetric analysis.

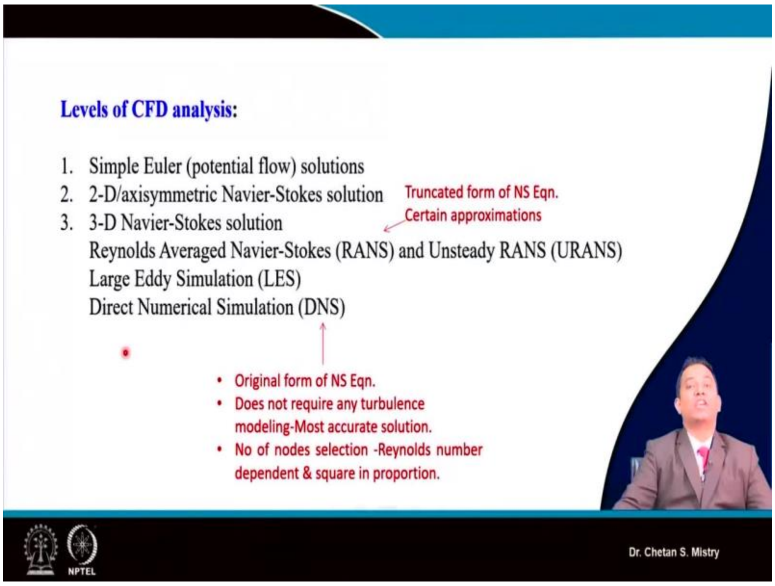
Then after they will be going with say detail analysis for say different blade rows. Suppose if you are having number of say stages to be on higher side, that's what will be analyzed based on data available for detailed design. Then they will be going for say 2D or three-dimensional design. Again, that's what will be developed and that what will be tested with the test rig as well as say engine test.

We need to be very careful, we have discussed what all are say the challenges for application of CFD. So, when we are doing our analysis that need to be validated experimentally. So, people they are assuming, they have presumed that CFD will be the replacement for this experimental testing that's what will take a long time.

As on today also, the results what we are getting from this computational analysis, that's what is little tricky and challenging and questionable until and unless that's what has been validated with the experimental results. These results are little tricky and of question. So, I am not saying that what all we are doing at this moment using CFD analysis is of no use; that is of use, but we need to have confidence in what all analysis we have done.

Being experimentalist, we are always having this as a challenge. And, that is the reason if you asked, I am more preferring towards say doing experimental analysis, and then maybe in order to understand detailed flow field study, we can go with this computational tool.

(Refer Slide Time: 26:17)



Now, if we look at there are different levels of CFD analysis. As we have discussed, very first, initially people, they were talking about simple Euler's or potential flow solution, that's what was a say, you know, initial approximation. Later on, people they went with say 2D and Axisymmetric Navier stokes solution. Later on, people they started talking about the three-dimensional Navier stokes solutions, where we are having RANS or URANS; we can say Reynolds Average Navier stokes equation, that's what we are solving.

Now, here, this is what is a truncated form of Navier Stokes equation, where we are doing certain assumptions. So, for the first cut design or the initial design, people, they are using RANS simulation that's what will give the quick solution to what analysis what we are doing with.

Later on, as per the expectation in terms of performance, people, they are going with say unsteady RANS simulation. Now, as we know, we have our limitations with the computational facility and the resources, that's what was say older story. Now, people they are moving towards higher end computational facilities, that's what will give more flexibility to go with say Large Eddy Simulations where we are analyzing large Eddy's and small Eddy's that's what will be giving the detailed flow features.

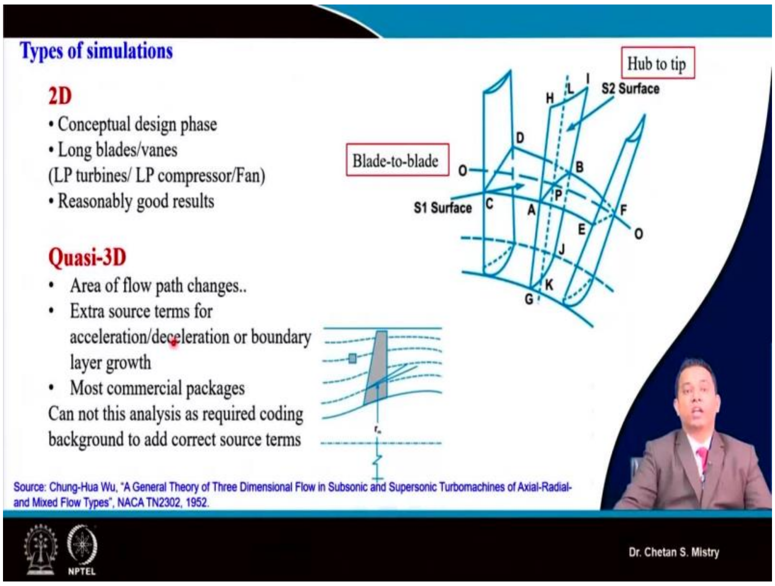
But the computational facility requirement for LES that's what will be huge. And, this is what required maybe weeks or months to analyze the simulation what we are talking in sense of designing the compressor. So, these tools cannot be used in order to finalize the dimensions, detailed flow field study, that's what can be done by using this Large Eddy Simulation or Direct Numerical Simulation.

Now, this Direct Numerical Simulation, that's what is taking into account for original Navier stokes equation. So, there is no need for having say turbulence model to be selected with, okay. And when we are having this kind of configuration, this is what will be giving more accurate results.

But at the same time, the number of nodes required, that's what will be the function of Reynolds number. And that's what is say in sense of say square. So, you can understand when we are analyzing say flow through axial flow compressor or the fan, Reynolds number, that's what will be in the range of $10^5\ or\ 10^6$, and that's what is need to be square.

So, that much computational facility that need to be there. So, people they are going with say initial analysis, at this moment with the small-small; maybe in future, we will be having those tools, that's what will be used in order to analyze the flow through axial flow compressor as well as say axial flow fans.

(Refer Slide Time: 29:34)



Now, types of simulation what we are discussing, that's what is say, two-dimensional simulation. So that's what is say for initial…say conceptual design phase, we can go with this 2D simulation. Mainly that's what is being used for the long blades or vanes. Suppose if you are talking about the fans or say LP compressor, we can go with this simulation where my flow we are assuming to be two-dimensional, okay.

Now, this is what will be give the reasonably good results for the initial first cut design. When we are talking about say analyzing our flow through turbomachinery, this is what was a concept, that's what was given by Wu in you can see in 1952. So, that's what was very initial stage of the computational fluid dynamics. And, you know, during that time, it was not that mature in sense of understanding.

Later on, people they realize this is what is of very much need, and that's what is initiation of development of computational tools. So, in order to analyze this part, what he has done? He has taken two surfaces; one surface, that's what is say S1 surface and S2 surface, we can say this S2 surface is nothing but it is from hub to tip and that's what we can say it is considered to be non-curvature.

So, this is what will be giving us details of how my flow that's what will be varying from say hub to tip, how my flow angles are varying from hub to tip. So, sometimes people used to say this as say, you know, the analysis of through flow analysis. This is what will be giving the details about through flow analysis and that's what is called S2 surface.

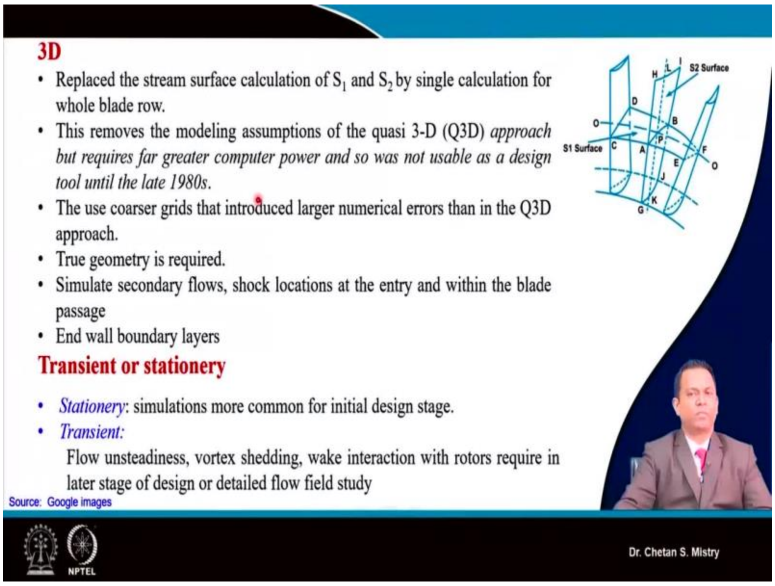
There is other approach, that's what is say S1 surface. So, you can see, like this is what will be giving the details about the flow between blade to blade surface. So, this is what is used in order to analyze what is happening with say within the blade passage, okay. So, this concept that can be used in order to modify our airfoils. How my flow does what is behaving on suction surface? How the flow is behaving on say my pressure surface? That kind of analysis that can be done. So, this is what is defined as a Quasi 3D approach.

Now, here in this case, say initially, when people they started doing their analysis, they were doing say, through flow analysis, based on this S2 surface, and they were doing the blade to blade analysis using this S1 surface.

Now, in order to analyze that, for our full flow field analysis, we need to have say additional source term to be added with. That's what will be talking about say acceleration or deceleration of flow, the growth of boundary layer. So, all this analysis that's what is required the understanding of coding. And, that is the reason why the corrected source term that need to be introduced in the commercial package, that's what is very challenging with.

So, when we are discussing about the flow for axial flow compressor, where we having per stage pressure rise to be very high, we know this is what need to be analyzed in a different way. So, my entry area and exit area will be different. This is what is been analyzed by considering say streamline curvature method.

(Refer Slide Time: 33:24)



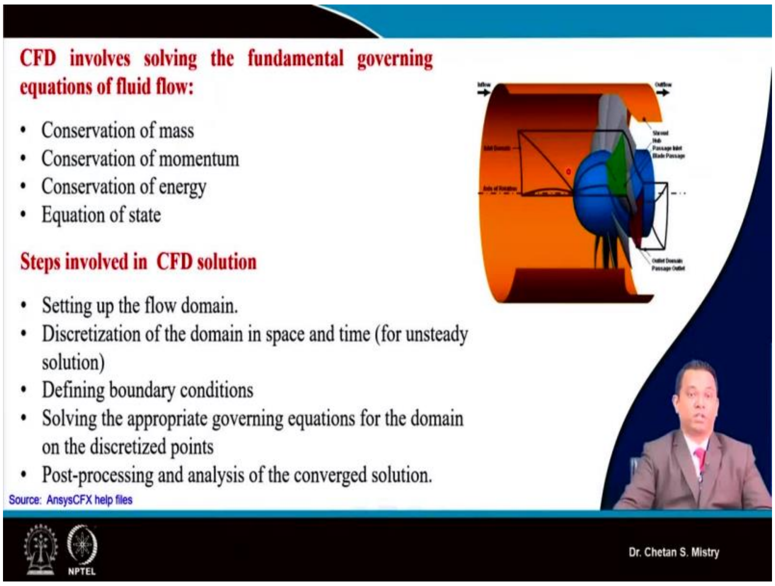
Now, let us see, this is what all we are looking at this moment in sense of three-dimensional analysis. So that's what will be the replacement of S1 and S2 surface calculation, okay. We are looking for the whole flow that's what is happening between the blade row. So, here in this case, that's what is removing the assumption of Quasi 3D approach, what we have discussed up till now; and that's what is the limitation till 80s. Now, with having say, advancement in computational facility, we are able to analyze the full three-dimensional flow through this blade passage.

So, use of coarser grid, that's what will be increasing the numerical error when we are going with say quasi 3D approach. This true geometry that's what is of need. The simulation of secondary flow, shock location at the entry within the blade passage, end wall boundary layer, those all analysis that's what is possible now with 3D solvers, okay. But the fundamentals, that's what has come from this Wu's analysis and later on, people they have modified the code in order to achieve detail flow field study.

Now, this flow field study that's what is people they are talking in terms of transient analysis and say steady state analysis or stationary analysis. So, this stationary analysis that's what is more common, when we are talking about the initial design iterations.

Later on, in order to understand the flow unsteadiness, vortex shedding, wake interaction between the rotor and the stator, all those detail study that's what can be done by using say transient analysis, okay. So, this all what we are discussing at this moment, it is giving the idea about the growth of computational fluid dynamics over the years.

(Refer Slide Time: 35:28)



Now, in order to have this analysis, we need to solve the fundamental equations. So, what all fundamental equations we are having, that's what is say conservation of mass, conservation of momentum, conservation of energy, and equation of state. So, these are all what we are talking of, that is nothing but it is the solver that's what is solving these equations in order to achieve the solution for the problem, okay.

Now, in order to do CFD analysis, we have certain steps that's what needs to be followed with. So, here if you look at, this is what is a fan. Now, in order to simulate or in order to do analysis for this fan, we need to make the flow domain first. Now, suppose if I am considering I am having, say more number of blades, or I am having larger diameter of the fan, under that configuration, my computational requirement will be huge. At the same time analysis also will be taking large time.

So, in order to simplify that we have approach, that's what is called periodic surface approach, okay. So, data, that's what is being transferred between these two domains, that's what is called periodic domain. We will be discussing all this in next lectures, but at this moment, you can say we need to set up initial domain. After that, we need to discretize this domain both in space as well as time when we are talking about unsteady analysis, we need to discretize in terms of times.

When we need to have the boundary conditions that what need to be defined with in terms of inlet boundary, outlet boundary, periodic boundary condition, no slip condition, free slip condition, all these things, we will be discussing in the next few lectures.

Now, after that, we will be having solving of these governing equations at different discretize point and based on that we will be achieving the solution for this particular domain. Then we need to analyze this result, that's what is called post processing and analysis of converged solution.

So, this is what is all we will be discussing in next few lectures. So, at this moment, this is what all we have discussed today will give you an idea of what exactly is a need of CFD analysis, and how we will be using this CFD analysis for the design purpose and later on for the performance assessment.

So, here we are stopping with the discussion of initial part for Computational Fluid Dynamics. In next lecture, we will be discussing how we will be doing this discretization, how we will be making our mesh, what all needs to be the boundary condition, what all care we need to take care of when we are doing our CFD analysis for say axial flow compressor, thank you. Thank you very much for your kind attention!