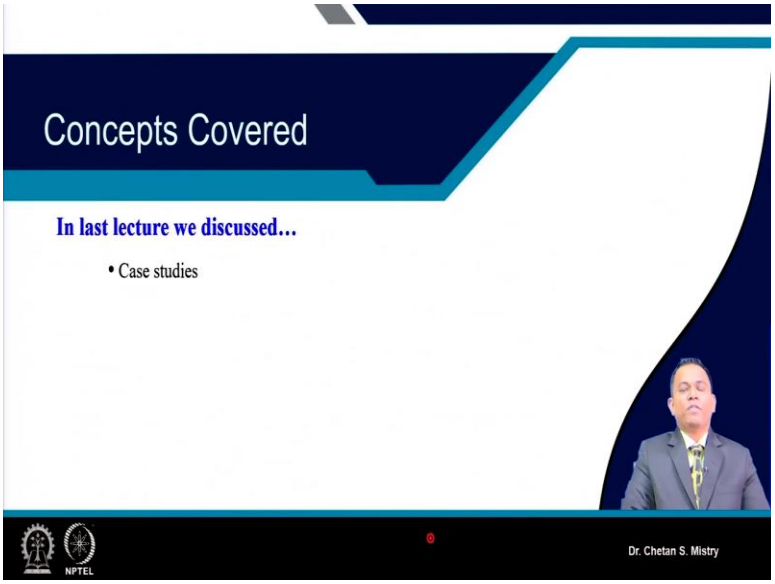
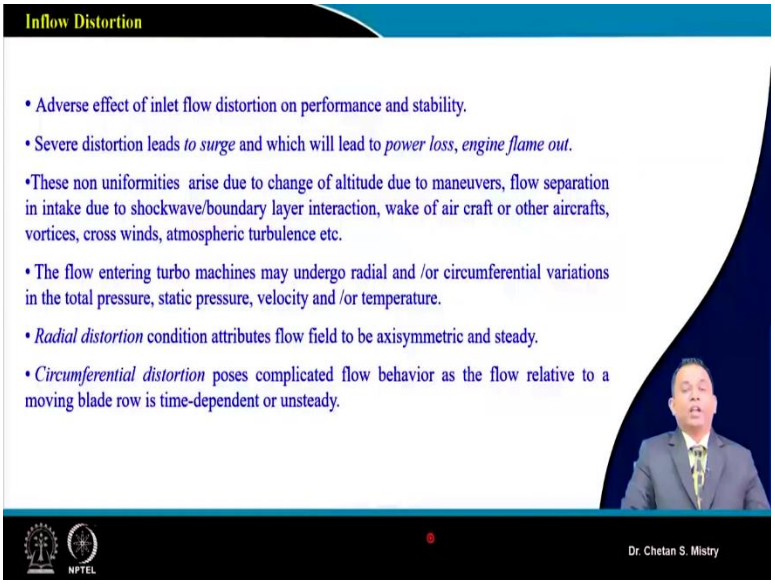
Aerodynamic Design of Axial Flow Compressor & Fans
Professor Chetankumar Sureshbhai Mistry
Department of Aerospace Engineering
Indian Institute of Technology Kharagpur
Lecture 69
CFD Application to Design and Performance Assessment (Contd.)

(Refer Slide Time: 00:29)



Hello, and welcome to lecture 69. We are discussing CFD application to design and performance assessment. In last lecture, we were discussing about different case studies, where we have done our initial design using say initial approach what all we have discussed in all 11 weeks. Now, based on that we have done our computational analysis, which is helping hand for modifying the design and later on after doing this design, it has been tested also.

(Refer Slide Time: 01:17)



Now, today we will be discussing some other aspects; say one of the aspects for aeroengine it is inflow distortion, rather saying aeroengine, same logic or same is a problem with all kinds of industrial fans also that’s what is called say inflow distortion.

So, this inflow distortion, it has adverse effect on the performance and stability of say compressor as well as fans. The severe distortion for say engine, it will lead to surge and it may possible we will be having power loss or maybe engine will go with flame out, that’s what is a complete failure of engine and that will be catastrophic failure of the engine as well as it will be the failure of whole aircraft.

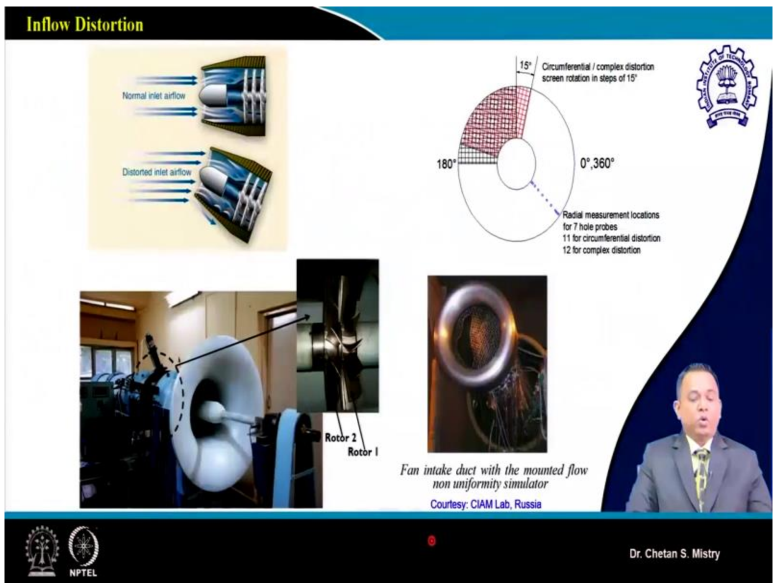
Now, this non-uniformity what we say it is because of change of altitude, mainly if we are talking about the fighter aircrafts due to maneuvers, the flow separation in intake due to shock boundary layer interaction, wake of aircraft or maybe aircrafts, vortices, cross wind, atmospheric turbulence etc.

So, when we are sitting in aircraft, pilot used to inform say tie up the seatbelt because of turbulence, this turbulence, that’s what is very danger phenomena it may lead to failure as we have discussed.

Now, this basically is happening mainly for say engines is because of radial or circumferential variation of total pressure, static pressure, maybe velocity or say temperature. So, change of any of this property, that's what will be bringing say inflow distortion, when we are talking about say radial inflow distortion, this is what is axisymmetric and steady, it is not that serious.

But when we are talking about the circumferential distortion, that is where we will be having flow which is in relation to the moving blade and that's what is the time dependent or the unsteady phenomena. Let us try to look at what all we are discussing and how it is being applicable here.

(Refer Slide Time: 03:54)



So, here if we look at, suppose say when we are flying at the cruise; so, the flow or air, that's what will be entering inside the engine in a normal way. But when we are discussing about say take-off and landing configuration or when we are doing our maneuvers, my flow, that's what is going smoothly on upper half or maybe lower half and it is getting distorted for the lower half.

Now, when this is what is going or striking on my blades, rotor blade for say compressor or say for fan then it will be subjected to say off design condition, means it is striking on a blade with different incidence angle. And, that's what will be giving the flow separation or the stalling of rotor or maybe stator and that's what will lead to the surge of whole stage.

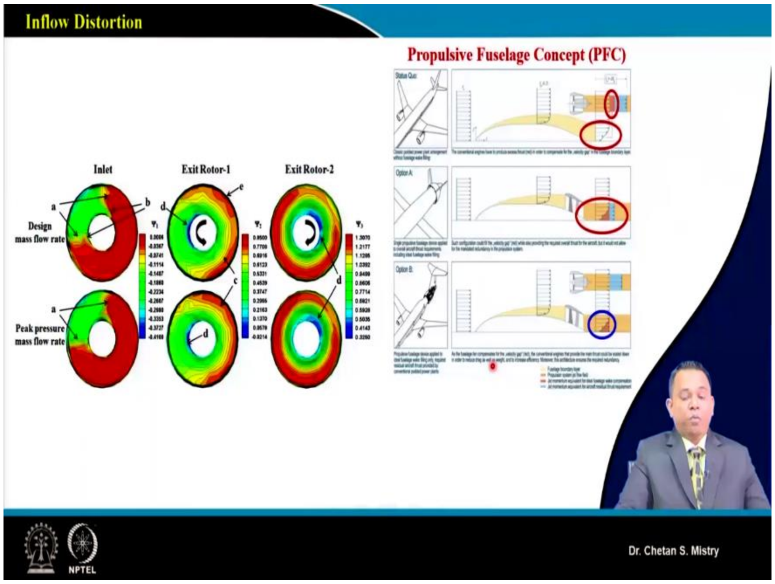
Now, in order to understand this phenomenon, for the same experimental facility, we have artificially generated say distortion by using this mesh configuration. So here, this is what is 90°

sector, that's what we have used for our study and that sector it is been rotated in the interval of 15°. And, in order to capture what is happening at the entry of my rotor 1 between both the rotors and at the exit of rotor 2, we have used 7 hole probe which will give more clarity in sense of change of flow field.

Now, in actual engine also, say this is what is experimental facility at CIAM lab, Russia. When we were doing our study for Contra rotating fan at low speed configuration, they people, they were exploring Contra rotating fan for high speed application and for that they have taken this is as the distortion screen.

So, basically in actual engine, they have done their measurement, how the total pressure it is varying. And, based on those results, they have generated this as artificial distortion, okay. So, you can understand how people they are exploring the possibilities for studying say distortion screen.

(Refer Slide Time: 06:30)



Let us see what all are the challenges here. So, this is what is representing say the presence of distortion screen at the entry of my rotor 1. Now, when this flow it is striking on rotor 1 at the exit of rotor 1 we can say my rotor 1 this is what is rotating in counterclockwise direction and this effect of distortion, that also seems to be rotating in the direction of rotation of rotor 1. And, here we can see the effect of the distortion at the exit of rotor 1.

Now, my rotor 2 it is rotating in opposite direction. So, we can say, the effect of this distortion that is rotating along in the direction of rotor 2, but at the same time what effect we are having in terms of inflow distortion, that's what is getting nullified here, do not go with this contour color, just look at the numbers, okay.

So, this is what is representing say design mass flow configuration, this is what is representing peak pressure mass flow configuration. With the interest of time, we will not be discussing in detail, but you can understand the use of Contra rotating fan it is giving benefit to nullify the effect of inflow distortion.

So, at the exit of my stage, we will not be having say any distortion effect, that's what is called distortion intolerant fans. This is what is a new thought process, lot of research and development activities that's what is going on. As we move ahead, we will see, for stage also, people they are exploring this kind of configuration. Now, the question is this all are the experimental results.

Now, in order to simulate this kind of configuration, single blade configuration may not help because we want to give say this as a circumferential distortion. So, we need to go with flow or say full flow domain simulation. So, all my rotor 1 and rotor 2 blades I need to configure and then this will be my inflow boundary condition and based on that we can do this simulation. So, this is what is one of the limitation we can say when we want to do simulation for inflow distortion and we know inflow distortion is very severe case or it is a severe configuration for the engines.

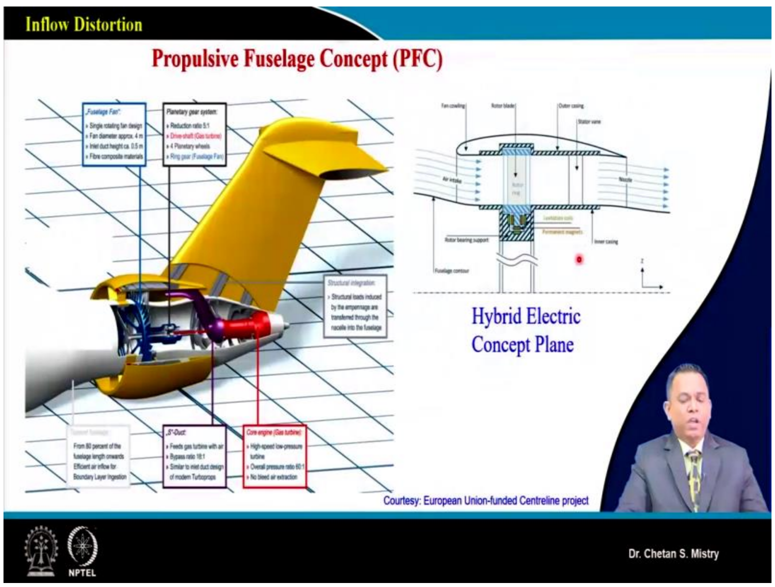
Now, there is one more logic or concept, people they are working on, that's what is called propulsive fuselage concept. So, here in this case, if we look at this is what is my aircraft body, we know from the nose of the aircraft, there is a growth of boundary layer that's what will be happening. We can say this growth of boundary layer, that's what is acting like a drag.

So, whatever thrust that will be generated by the engine most of that thrust, that's what is being utilized in order to overcome this drag. And, basically, because of that the total thrust available, that's what will be requirement for total thrust will be higher and that's what is impacting on say specific fuel consumption.

Now, people they have explored the possibility putting the engine on say the rear side or near the trailing edge of the aircraft like this, that's what will be acting like, you know, minimizing the effect of boundary layer growth in this configuration. Again, the thrust requirements or the thrust generated by this engine, it is not sufficient or it is less.

So, in order to ride or get to work this kind of limitations, people they have proposed with say engine as well as this is what is called say propulsive fuselage unit that's what has been placed here. So, the effect of drag or say boundary layer, that's what has been getting nullified by using this kind of fan configuration, and the thrust, that's what will be generated by the engine, that's what will be sufficient for the running case.

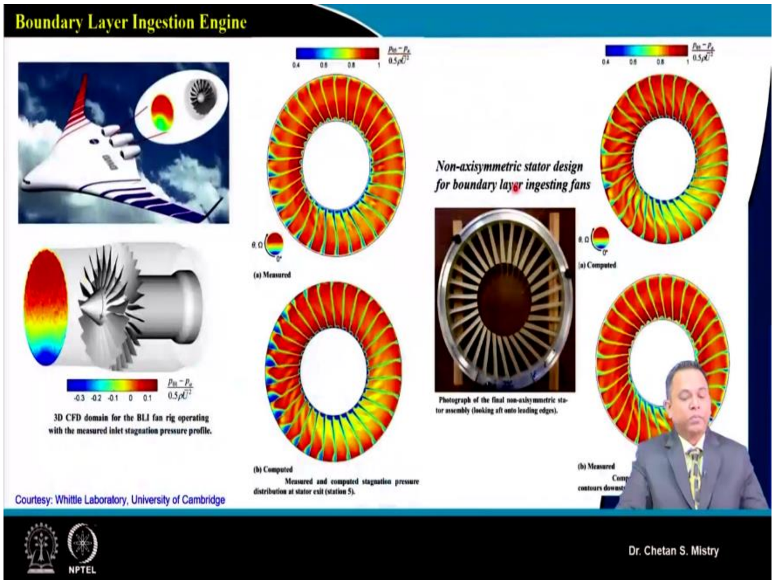
(Refer Slide Time: 11:19)



Now, if we look at carefully, this is what is say European Union funded Centre Line project. So, in this case, this is what is a fan; now, again, this is what is a fan it is a reason why this is what is of our interest.

Now, this fan, that's what was run by say gas turbine engine, and we can understand this fan it is being used as a propulsive device in order to minimize the drag effect and in order to improve the specific fuel consumption. Later on, they people, they have explored using say hybrid electric configuration, where they are having say BLDC motor, that's what is being used in order to rotate this fan, okay.

(Refer Slide Time: 12:08)



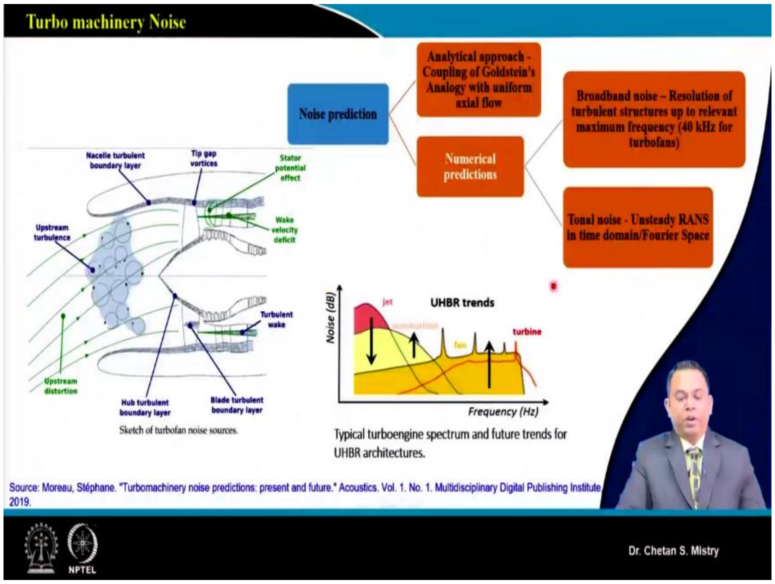
Now, this is what is ongoing work at say Whittle Laboratory, University of Cambridge. Now, future aero-engines or aircraft, that's what will be a wide body aircrafts. Now, for that the engines will not be installed below the wings, the engines will be installed here. So here, you can say this is what is say coded kind of configuration.

So, engine will be placed here, like half of the engine that's what will be buried inside the body…inside the body. So, what will happen? The flow, that's what will be coming out from this nose, it will be moving downside. So, we can say, it will be having the growth of boundary layer. And, that is the reason we will be having distorted flow, that's what will be entering inside the engine.

So here, we can see this is what is a distorted flow, that's what is going inside the engine or that's what will be striking on the fan. Now, we are looking for the configuration where we will not be having any effect of this distortion downstream. Because if this is what is affecting the performance of my fan, that's what will lead to stall and surge kind of configuration.

So, for that they people, they have developed say professors and say University of Cambridge, they people, they have developed this configuration of the stator. If you look at carefully, say the shape of the stator they are being modified in order to address the issue of inflow distortion. So, this is also one of the application of CFD for the future engine development.

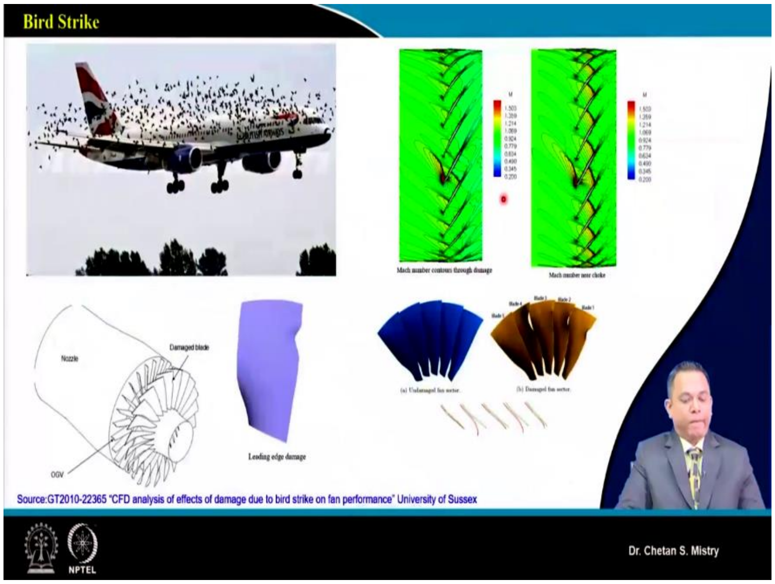
(Refer Slide Time: 14:11)



The major issue, that's what is coming now, in order to meet the requirement of a ACARE, as per say ACARE 2020, is noise. So, noise also is coming into the picture when we are talking about the design. So, here in this case, if we look at, this is what is say inflow turbulence, we will be having growth of boundary layer, we will be having tip vortices, wake, that's what will be coming out, we will be having stator that's what is a stationary component.

So, under this configuration, if we look at for ultra high bypass ratio engines, we will be having fan, that's what is contributing more noise. So, now in order to design the fan, that's what will be giving say less tolerance or maybe having the issue of noise that can be done by using this computational tool. But here, we are looking for more challenging development activities in terms of development of new code, that's what will be helping in order to minimize say noise.

(Refer Slide Time: 15:29)



Now, this is what is a very challenging aspect, what we say is a bird strike you can see here. Now, when the bird, that's what is coming in front of the engine, that will suck inside, it will damage the rotor blades for the fan, maybe for stator blades also, and maybe it will give the surge kind of configuration and immediately we will be having flame out.

So, now engine design companies they are looking for say doing simulation for say design and development of these fan blades. So, here if you look at, these are the damaged blades, and this is what is representing how my flow that we will be having.

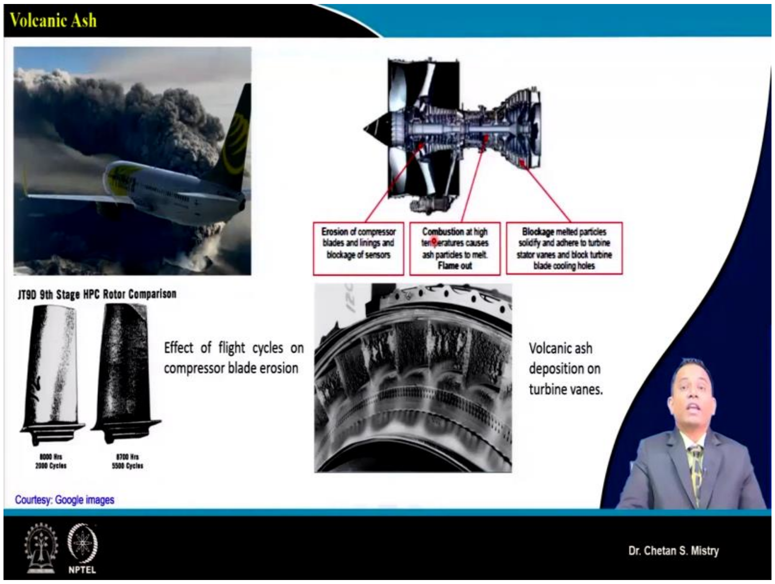
So, under this kind of configuration, the use of computational tool that will be helping a lot. But at the same time, the simulation for such kind of blades, you can understand, the type of say modeling requirement, meshing requirement, then computational power requirement, all those they are very challenging.

(Refer Slide Time: 16:43)

Now, this is what is one of the major challenge that's what is called say icing issue. So, when these aircrafts, say commercial aircrafts, when they are flying at high altitude, maybe in the range of 41,000, or 40,000 feet height, under that configuration, this ice particles, that's what will be going inside the engine. Now, when they are going inside the engine, they will get deposit on say rotor blade as well as stator blade.

So, this deposition, that's what will give off design kind of configuration, because my blade shape that will be changing. And, that's what will lead to say surge kind of configuration for the engine. Many times, because of say deposition of my ice particles on the blade, later on as the rise of temperature that is happening in a later stage that's what will give say striking of this particle on say downstream LP compressor or HP compressor even that's what will be going in combustion chamber. So, this is also one of the challenging problem. So, how to do simulation for this kind of configuration.

(Refer Slide Time: 18:10)



Volcanic ash, so when we are having this volcanic part, this active, this ash particle, that's what will be entering inside the engine. Now, when that's what is entering inside the engine that will get deposited. So, here this is what is representing for say turbine blade and this is what is representing say HP compressor blade. So, once we are having this particle that will be deposited on the surface, now, whole flow physics, that's what will be changing. So, now, we are looking for the course, which will be helping in order to simulate this kind of configuration.

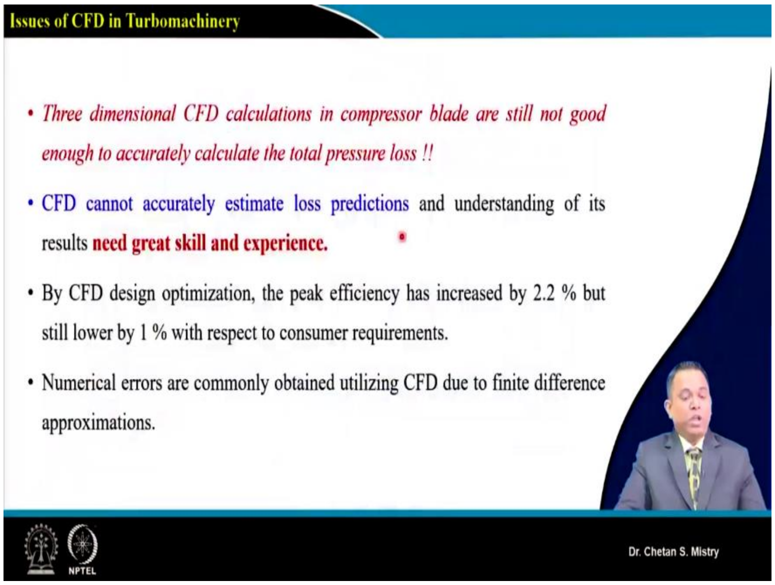
(Refer Slide Time: 18:56)

Now, this is what is project FULLEST. So, up till now, what all simulations we have discussed, up till now, what all CFD we have discussed, that's what was limited to say rotor-stator configuration. We have not discussed say when we are talking about multistage configuration, the whole simulation, that's what will be required whole lot of time, computational power and whole lot of understanding, then we need to take care of interface and all those terms. This is what is say LES analysis, what this companies and universities combinedly they have done.

So, here if you look at carefully, this is what is representing how the wake, that's what will be coming out from the rotor, it will be striking at particular location for the stator and this is what is downstream how the flow is behaving.

Now, we are looking for this kind of simulations. This is what is computationally very expensive. When we are talking about LES, we have discussed in initial lectures, when we started discussing CFD, we have discussed these aspects. Say, my mesh size requirement, my computational power requirement, my time requirement for the simulation, that's what will be huge.

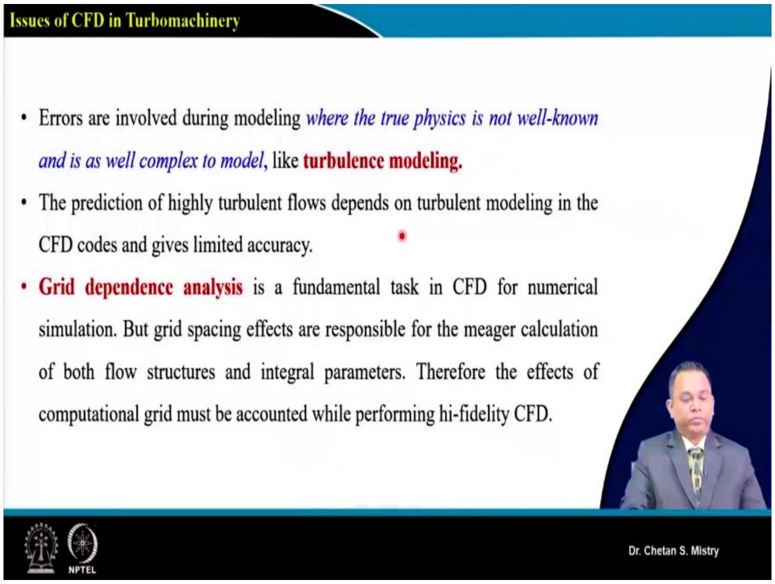
(Refer Slide Time: 20:14)



So, in overall if we discuss, there are certain issues with CFD analysis. So, three-dimensional flow what we are looking at for the compressor still as on today, the results are of question mark. So, these codes are unable to predict the total pressure loss. At the same time, in order to have the accurate prediction of the performance, that's what is required whole lot of skill and experience.

Next, if we are simulating our say stage and we are claiming say improvement in efficiency of 2.2%, still that consumer expectation is on higher side. Then, numerical errors we are finding, that's what is because of our finite difference approximation.

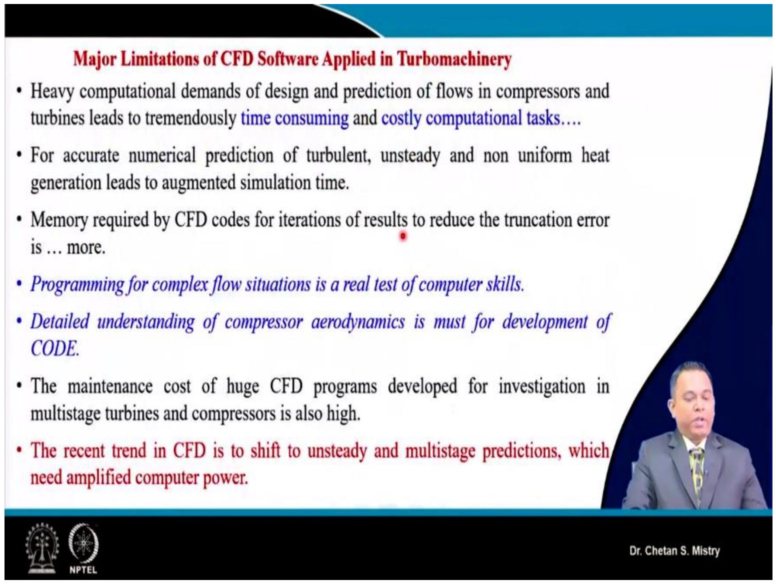
(Refer Slide Time: 21:15)



Now, these errors, we are unable to understand the physics, actual physics. And, until and unless we are having an understanding of actual flow field, we are unable to select particular turbulence model. And, that's what will lead to give wrong kind of results or it is not giving what we are expecting in terms of accuracy. We need to go with grid independent study what we have discussed.

Now, this is very important aspect when we are looking for say flow structure and we are looking for internal flow parameter calculation. The effect of computational grid must be accounted while performing say high fidelity CFD. Suppose, we are talking about say LES simulation, where this grid independent study, that's what is required very fine mesh in order to capture those eddies.

(Refer Slide Time: 22:31)



Now, major limitation when we are talking of, say we are looking for say design, okay. And, this design, that's what if we are going with the CFD analysis, we need to go with number of iterations and that's what is a time consuming and it is computationally very expensive task. For accurate prediction of turbulence, unsteady or non-uniform flow, that's what is required augmented amount of simulation time. We can say, we are assuming our turbulence intensity for the initial design case, but later on when we are going what say performance assessment, we need to be very careful; as we have discussed, suppose say inflow distortion, that's what we are simulating.

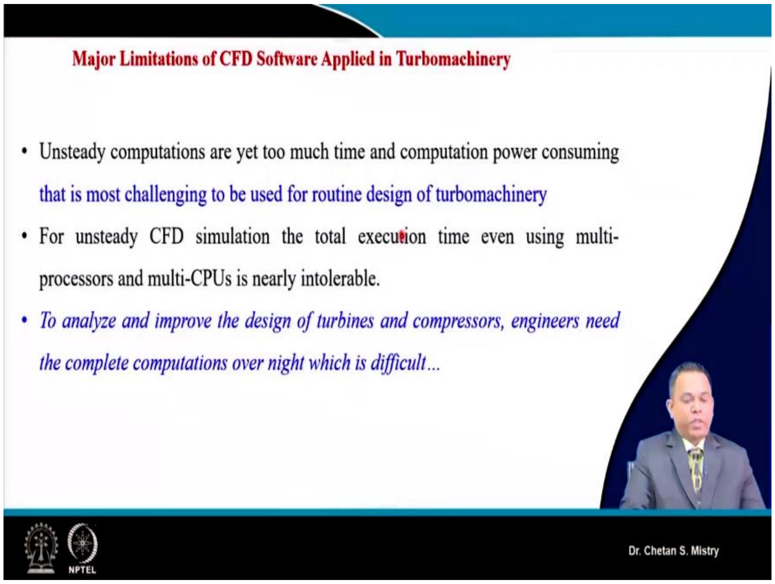
Now, in order to understand this inflow distortion, these are all parameters they are very important and you have no other option than doing the experimentation. Now, people they are developing their design of stages which are say distortion intolerant, so, initial design stage only they want to do this simulation. Now, this is what is a big challenge.

Now, memory required for the CFD code for the iterations, that's what will can be reduced with the truncation error, that's what will be increasing; suppose say we are looking for more accurate readings, then my time requirement, that's what will be huge, that my error in simulation also will be higher.

Now, programming of complex flow is really very challenging task. Until and unless we are having detailed understanding of aerodynamics it is very challenging to develop the code because this is what requires a whole lot of understanding.

So, coding for turbo machinery or say for application of axial flow compressor and fan, that's what is very challenging tasks. So, maintenance cost for huge CFD program, that's what is requirement for multistage configuration will be very high. So, recent trend for CFD is moving towards say unsteady simulation and multi stage configuration kind of simulation, that's what is required very huge amount of computational power.

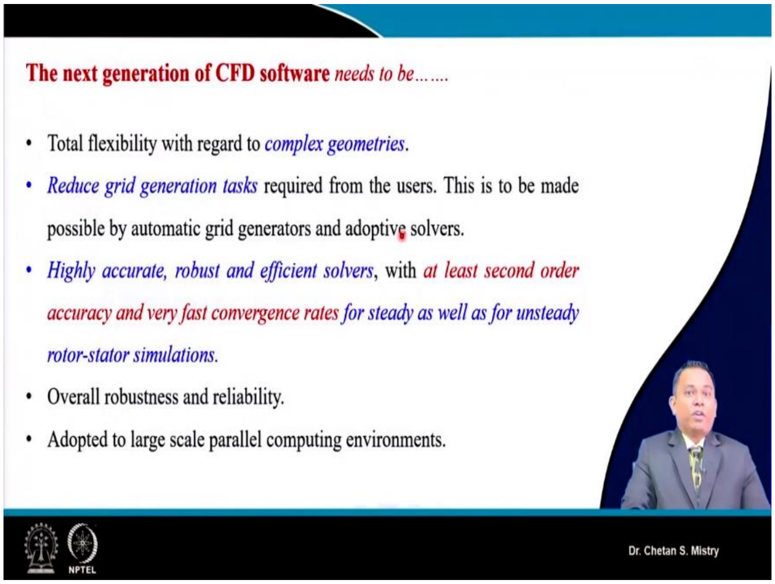
(Refer Slide Time: 25:06)



Now, unsteady simulation, that's what is say time consuming affair as we have discussed. So, that we cannot do for initial designing stage. So, initially when we are doing our design that time we are going with our steady simulations, when we are going with say detailed performance assessment, then only we are looking for unsteady simulations.

But the trend that has changed. Now, people they are looking for unsteady simulation at the designs stage only and that's what is required huge amount of understanding, huge amount of computational power as well as time. To analyze and improve the design, the engineer needs to go with say complete simulation at the initial stage, that's what is very difficult and challenging as we discussed.

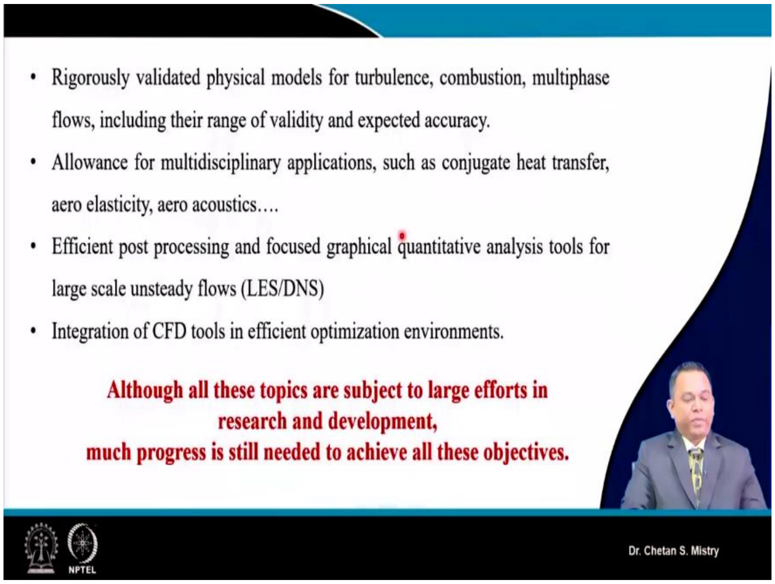
(Refer Slide Time: 26:00)



Now, what all we are looking for say future safety software's to do? It says, like total flexibility regard to complex geometry. So, we have seen, suppose say bird strike kind of configurations or say maybe when we are looking for three-dimensional shape of our blade, for that we need to have very good control; reduce the grid generation task as per the requirement, because whole lot of time, that's what is going for the grid generation.

So, now automatic kind of configuration, that's what is next a demand. Highly accurate, robust and efficient solver, with at least second order accuracy, and very fast conversion rate for both, steady and unsteady simulation, that's what is of demand. Now, overall robustness and reliability, that's what is also equally important. So, adoption for large scale parallel computing environment that's what is a future.

(Refer Slide Time: 27:05)



Now, in order to go with say design stage or performance assessment stage, we need to have validation of the computational results and that's what is demanding for whole lot of experimentation, there is no option to that part.

Now, when we are talking about the multidisciplinary application, say bird strike kind of configuration or flutter kind of configuration, or maybe noise kind of configuration, where you are looking for say simulation, that's what will be of different tasks, it is a multi-physics kind of configuration.

Now, efficient post processing and focus graphical quantitative analysis tool for the large scale unsteady flow. So, now, after doing this simulation or solving our problem, the post processing also is equally challenging.

Now, we are looking for those kinds of tools, which will be helping us in analyzing those unsteady simulations or LES or DNS simulation. People, they are demanding for CFD as optimizing tool so, that's what is a next challenge for the coders or commercial package developers. Companies, they are working on these aspects; many companies, they are having their own optimization tool, which helps them to reduce the number of iterations. So, all these topics are subjected to large efforts in terms of research and development activities. Much progress, that's what is still in need in order to meet all these objectives and requirements.

(Refer Slide Time: 28:57)

Now, let us see what all people they are working throughout the world in terms of say axial flow compressors as well as fans. So, now, recent trend, that's what is say development of sixth generation aircraft for say Aero Engines or fighter aircrafts, where people are talking about variable cycle engine, in which we will be having our stator to be variable or that's what will be of different requirement. So, the design for such stage, they are really challenging.

Next, that's what is say contra-rotating concept for compressors, ducted and unducted fan; people, they are talking about say axial centrifugal compressor stage, distortion tolerant say rotors, tip insensitive rotors. Now, this is what is of different flavor. We know, there is a tip leakage flow, that's what prone to happen because of relative clearance between rotor and casing. Now, the demand is those kinds of fans or say compressor, that's what will not be having any effect of say tip clearance. So, that's what is demanding for whole lot of understanding.

Then tandem bladed, splitter blade, aspirated rotor and stator configurations, mixed flow turbomachinery, stall inception, active and passive flow control for performance improvement and the noise, application to UAVs, propulsive system based on say electric fans, we have discussed about this part; unducted fan for aircraft, ultra high bypass ratio fans, fan and noise suppression, engine icing study, special application axial flow compressors and fans, FOD ingestion, multi flow application.

So, all this kind of research and development activities, that’s what is going on throughout the world at this moment. So, if you look at this blue color highlighted portion, that’s what is ongoing work at IIT Kharagpur with my team.

Now, we need to understand until and unless we are having background of initial design, we cannot move forward, that is one case. Second, already designed things, that’s what is available with you in which you will be going with the modification in order to meet the requirements.

But there also you need to have your understanding of aerodynamic design. So, there is no alternative to have detail understanding of design, because that is where you are having a whole lot of flexibility for future requirements. So, all what we are discussing at this moment, that can be addressed with say systematic aerodynamic design of compressors and fans.

So, this is what is end of our week 12 in which we have discussed about the CFD application for design and performance assessment. I am sure, this will give you detail understanding how we will be using our computational tools for design and performance assessment. Understand one thing, because of our time constraint, and this course is not related with the CFD, we are not discussing so many aspects here.

Those who are really interested to make their future or those who are already doing their CFD analysis, if they are looking for detail understanding, maybe just go with recent research papers and that’s what will be helping them in detail understanding for application of CFD. So, here we are stopping with this course with say end of week 12. Thank you. Thank you very much. I hope you will be having great understanding for design as well as the use of the CFD. Thank you!