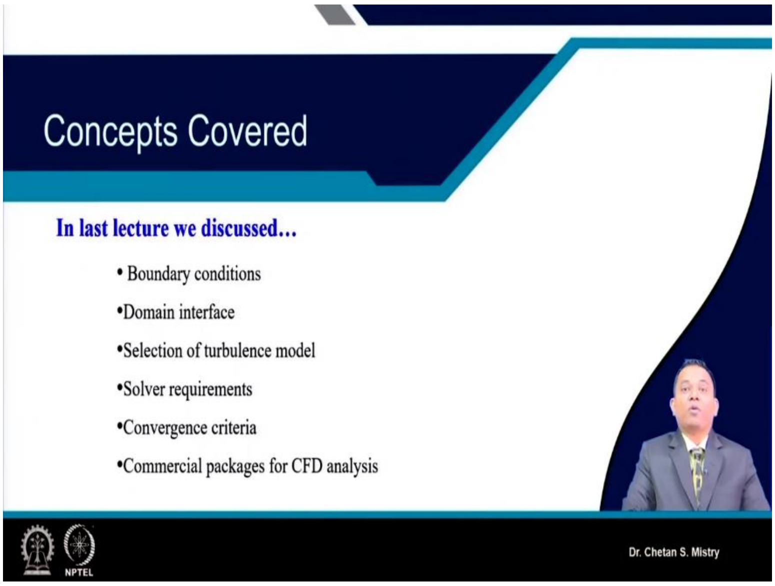
Aerodynamic Design of Axial Flow Compressor and Fans
Professor Chetankumar Sureshbhai Mistry
Department of Aerospace Engineering
Indian Institute of Technology, Kharagpur
Lecture 68
CFD Application to Design and Performance Assessment (Contd.)

Hello, and welcome to lecture 68. We are discussing about CFD Application to Design and Performance Assessment.

(Refer Slide Time: 00:37)



So, in last lecture we were discussing some of the important aspects for the CFD analysis application to axial flow compressors and fans. We were discussing about different boundary conditions, if we recall, we were discussing about inlet boundary condition, outlet boundary condition; wall condition, basically casing, hub, blade, that's what we are considering as a no slip condition, then we were discussing about the periodic condition.

We also started discussing about the domain interfaces where we have discussed about mixing flow approach and we have discussed about the frozen rotor approach. And, we realize in order to have what exactly we are looking for, say, are we looking for detail flow field study, that time some of the averaging that's what we need to avoid.

So, data that will be exchanged between rotary domain and stationary domain, that's what has been interface by using frozen rotor approach. We have seen how the data, that's what will be transferred from rotating domain to say stationary domain. And, here in this case, we have realized the clocking effect, that's what is coming into the picture, that's what is basically relative position between two domains.

So, we need to be very careful when we are using one of the approach for say solving of stage; maybe, say rotor-stator or maybe rotor-rotor configuration for contra rotating fan. Then we were discussing about the selection of turbulence model, we have discussed about BA model, we have discussed about SA model, and we realize for the initial stage of design, where we will not be having flow separation, there we can go with those turbulence models.

But we know when we are doing our initial design, we are not having clarity, how exactly our flow will be behaving on a suction surface and on pressure surface, as well as what will be the flow behavior in the tip clearance region. And, that's the reason why, even at the design stage also, people they realize we need to go with say $SST\ k-\omega$ model, which will take care of coriolis component and that's what is best model what people they are opting for say analysis purpose.

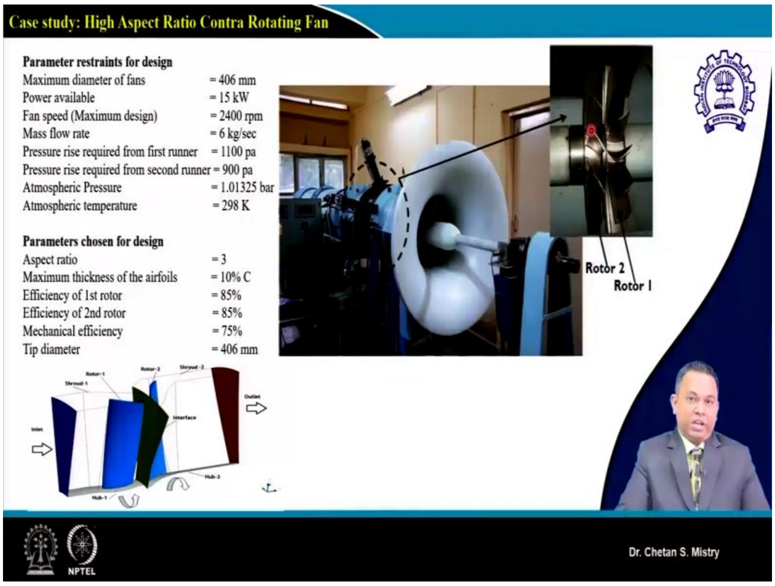
Recently people, they started talking about $\gamma-\theta$ model, that's what is giving idea about the transition of flow from laminar to turbulent, and we know that flow phenomena, that's what is very important. Some of the important flow feature we are missing when we are going with some other turbulence models.

So, selection of this turbulence model is equally important, when we are talking for say the accuracy of our analysis. We were discussing about the solver requirements, where we have discussed say compressible solver, incompressible solver, those solvers also can be corrected as per the requirement like we can go vice versa. Then we started discussing about the convergence criteria.

So, exactly in sense of what parameters we are checking with or what analysis we are doing that need to be observed. So, we will be putting some observed parameter with some RMS value or some average value, that's what will be continuously under observation. And, that's what will give you the confidence in sense of the simulation, that's what we will be finishing or we already have finished with. So, that's what we need to realize here. Then we started discussing about different commercial packages for the CFD analysis.

Now, the thing is, after having all this brief idea about the CFD and CFD tools and their processing, we must get some confidence in terms of using this, say, maybe understanding of CFD or say using our commercial packages. So, today we will be discussing about different case studies over the years what we have explored.

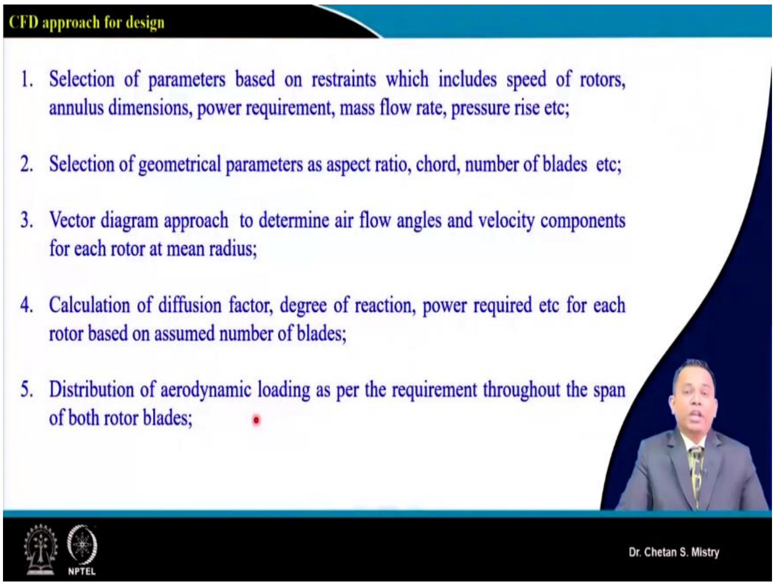
(Refer Slide Time: 05:14)

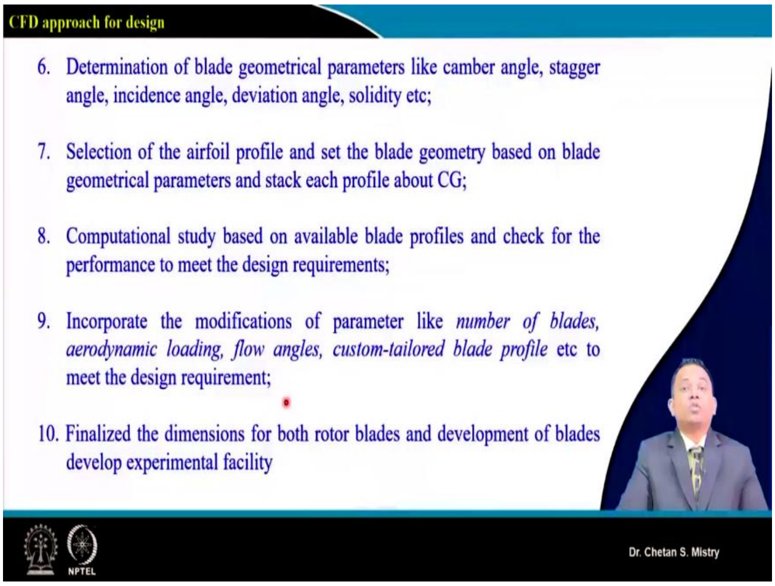


So, let us take very first case study, this is what is a case study for contra rotating fan, this is what I have explored when I was doing my PhD at IIT Bombay. And if you recall when we were discussing the design of contra rotating fan. Similar kind of configuration we have discussed, here we are having two rotors, if you look at carefully, say we are having rotor 1 and rotor 2. This rotor 1, that is rotating say in clockwise direction and rotor 2 that’s what will be rotating in anti-clockwise direction.

So, now, in order to do design for this contra rotating tool or say contra rotating fan, we need to use the computational tool. So, for that purpose we have used ANSYS CFX for say solving this design or design problem. So, before finalizing we need to do or we need to go with number of iterations. So, this is what will give you some confidence, some idea how exactly we need to use this tool after learning say meanline design or say preliminary design what all we have discussed in last 11 weeks.

(Refer Slide Time: 06:33)





So, say very first step, that's what is coming as is a selection of parameters based on the restraints which includes, say speed of the rotors, annulus dimension, power requirement, mass flow rate and pressure rise. So, actually we were available with say bellmouth, that's what was having say inner diameter of 406 mm, we were having motors, they were having capacity of 15 kW and based on that we are having some constraints that need to be incorporated for the design.

So, maximum diameter, say rotational speed, power, they three were the initial restraints for our design. Then we have selected say geometrical parameters like aspect ratio, chord, number of blades, etc. So, this thing what we are moving with say meanline design, the same way this is what has been done. So, vector diagram approach for determination of flow angles, different velocity components at the mean radius for both the rotors, calculation or the observed parameter what we learn, say diffusion factor, degree of reaction, power, based on what all number of blades we have selected with.

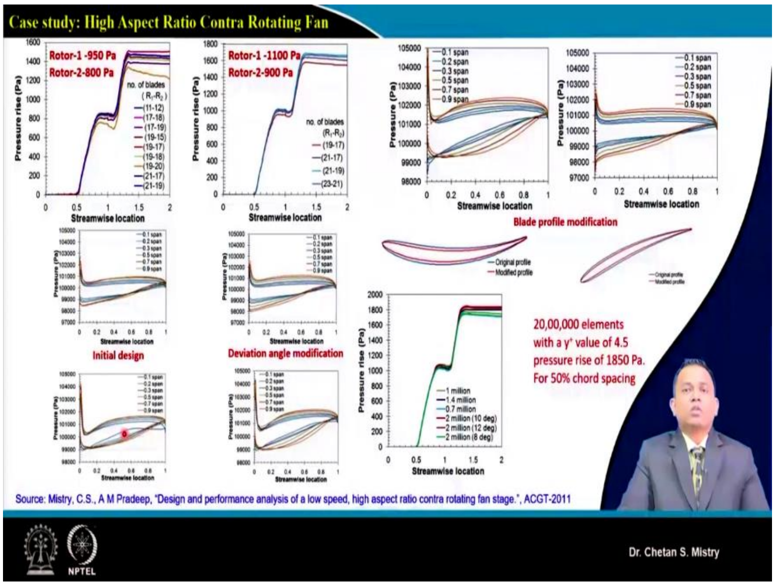
Distribution of aerodynamic loading as per the requirement throughout the span for both the rotors and for current design or what design we are discussing. We have taken say highly loaded rotor, rotor 1, we were expecting our pressure rise to be 1100 Pa for rotor 1, and for rotor 2 it was 900 Pa. So, total expected pressure rise from this contra rotating stage, it was 2000 Pa.

Now, after doing all this, say meanline calculation, we have calculated our camber angle, stagger angle, we have assumed our incidence angle, we have corrected our deviation angle, number of blades, solidity, chord, all those parameters that's what is coming into the picture.

Then next step, that's what is with the selection of airfoil, that's what was stacked about CG, we can understand both the rotors are rotating. And, that is the reason why we need to stack both the blades or all airfoils for both the rotor blades about CG. Then based on say available profile, say initially, we have taken our C4 airfoil for the design and later on as per the requirement, we have modified.

So, it says incorporation of modifications like number of blades, aerodynamic loading, flow angles, custom tailored blade profile, etc., in order to meet the design requirements. So, this is what is a systematic way of doing the design. Now, this design modification, that's what was done by using computational tool and that's the reason why specifically I am putting this in a discussion. Now, once we have finalized all of our dimensions, later on, we made our experimental setup, for both the rotors.

(Refer Slide Time: 09:58)



Here, in this case, initially, we have started doing design, which say expected pressure rise of 950 Pa and 800 Pa. And, we have understood like how exactly my aerodynamic loading, that's what is happening on say pressure surface and the suction surface for say both the rotor. So, this is what is representing at different span location for rotor 1 and this is what is representing say aerodynamic loading at different span for rotor 2.

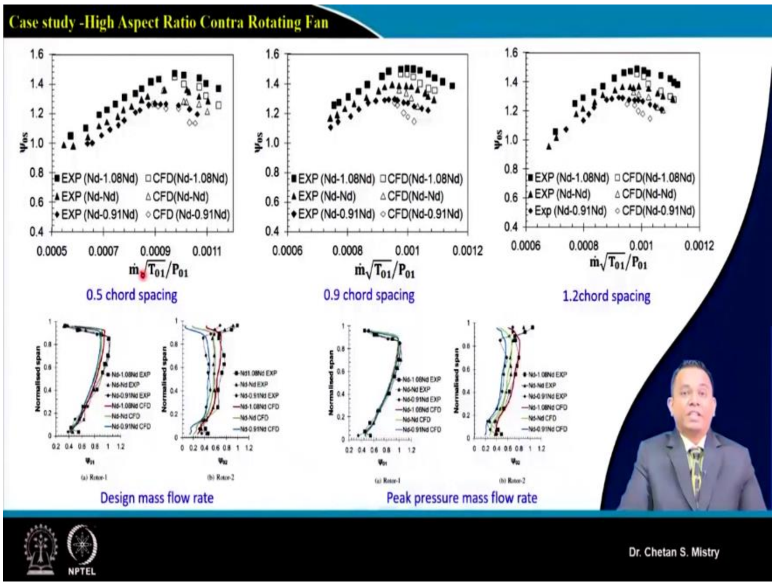
Then slowly we have increased our aerodynamic loading as per our expectation to 1100 Pa and 900 Pa. So, when we have done this part for that, we have initially fixed our number of blades based on number of iterations. Then we need to go with say for this particular aerodynamic loading, whether this number of blades will be coming to be as per requirement or not.

Let me tell you, here in this case, our blades are high aspect ratio blade. So, we can understand the height of the blade is large and chord is smaller, so we can say 135 mm that's what was the height of the blade and 45 mm was the chord of both the rotors. We can understand this 45 mm and within this blade passage to do diffusion, that's what is very challenging. But that's what is a challenge for the design and that's what we have opted for.

So, with this we have done number of iterations in sense of modifying the deviation angle, with modification of deviation angle we can see the aerodynamic loading for both rotor 1 and rotor 2 at different span, that's what was getting changed. Still, we were unable to achieve what pressure rise we were expecting, later on those airfoils they were been modified.

So, you can see, this blue color one, that's what was original C4 airfoil. Later on, we have modified by shifting say maximum thickness point as well as reducing the thickness of this airfoil. And by doing so, we were able to achieve the pressure rise as per our expectation. So, by having this kind of initial study, by having say initial analysis, we will get the confidence about the design, again let me put you point here, this CFD is not a design tool, this is what is for design verification, okay.

(Refer Slide Time: 12:37)



Now, once this is what has been done, we have developed our experimental setup and, here if you look at, we have discussed earlier what all are the benefits of contra rotating fan where we have discussed about the partial stall and full stall configuration, I will not be repeating here. And, here if we look at, say this filled points they are representing say experimental results for different spacing between both the rotors.

Now, in design, at the design stage, when we are doing preliminary design stage, that time this spacing will not come into the picture, but when we are doing our analysis or when we are making our both the rotors then what need to be the distance or axial spacing between these two rotors, that’s what is very important, okay.

Basically, that’s what will be deciding your aerodynamic performance, that’s what will be deciding in terms of say your expected noise as well as the length of your stage. Suppose say we are designing say multi stage configuration for contra rotating application, that time the spacing between these two rotors, that’s what is very important. So, this is what is representing say 0.5 chord axial spacing, 0.9 chord axial spacing and 1.2 chord axial spacing.

Initially, when we started doing design, that time we have assumed our axial spacing to be 50% of rotor chord 1. Later on, by experimentation we realize 0.9 chord or 90% chord of rotor 1, that's what is a spacing it is giving improvement in terms of performance. And, that's what was later on being used for all analysis purpose. This is what is representing the validation.

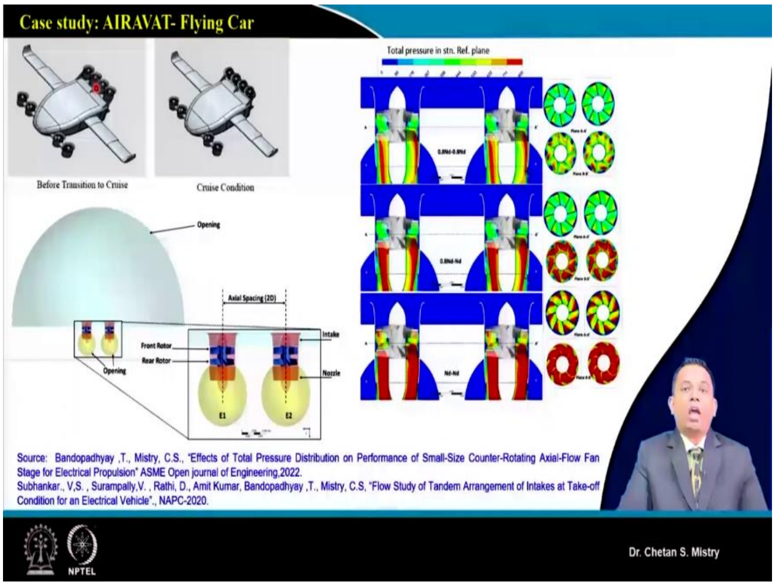
So, if you look at these unfilled points, they are representing say CFD analysis, if you observe carefully for the high mass flow rate configuration, it is not matching well, it is having certain amount of discrepancy, but when we are moving towards say peak pressure point that time it is matching well for all different speed combinations. Later on, after having experimental result, say as we have discussed your inlet condition you can take say variation of total pressure.

So, having say experimental results, we have considered that profile as input, say total pressure input. Same way, we have used seven-hole probe for the measurement of velocity with that boundary condition also we have done our simulation. And, this that's what was being used in order to validate our say CFD and experimental results.

So here, this black color one, that's what is representing our experimental results and this that's what is representing our CFD results. So, if you observe carefully, like for rotor 1, when we are having say design mass flow configuration, that time it is not giving much comparatively good agreement between experiments and computational work, but when we are talking about say for rotor 2, it is matching well. At the same time, as we have discussed here, say during peak pressure configuration or say peak pressure mass flow rate, it is giving very good agreement between experimental and computational results.

Now, this is what is a question what we say in terms of limitations of your computational fluid dynamics or still it required lot of digging inside in order to achieve the validation of experimental as well as computational results.

(Refer Slide Time: 16:46)



Now, based on the confidence what we build by for say contra rotating fans, this is what is a second work, that's what is going on at IIT Kharagpur, it is project AIRAVAT. We are developing a flying car, say prototype model of 10 kg, that's what is in progress. In order to have that, this is what is our proposed model in which we will be having say ducted fans on the front side, we will be having ducted fans on the rear side.

Now, in order to have say lightweight, we have discussed these aspects when we were discussing the industrial fans, rather having big size single fan, it is preferred to go with number of fans or say maybe going with say number of motors. So, that's what will be giving benefit in terms of say weight as well as what we are talking about the endurance, and motor capability. And, that is the reason why we have gone with this multi fan configurations.

Now, here when we are flying at the cruise condition, say this is what is say vertical take-off kind of configuration where we are having say telescopic wing, that's what will be giving the benefit in terms when we are going for say cruise, when it is on road it will not be having say extension. So, during cruise condition, rear fans, that's what will be getting tilted, and that's what will be giving the forward movement.

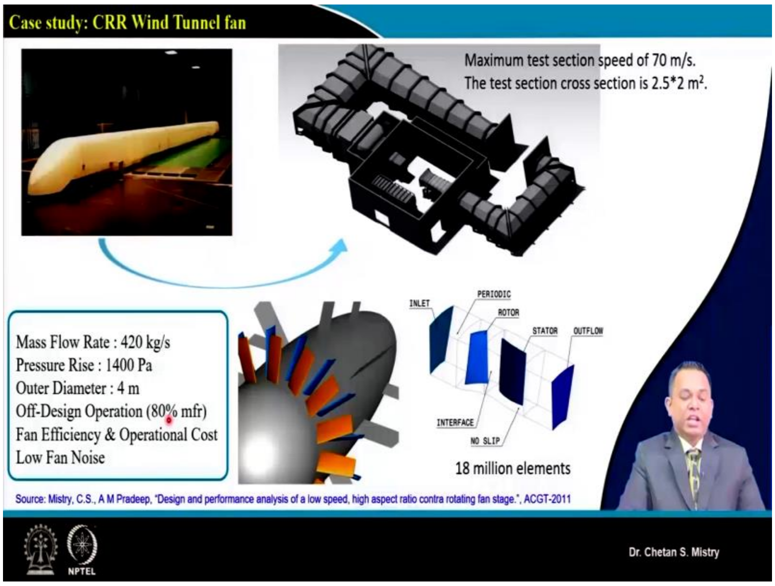
Now, the question, there are many questions for such kind of proposals when we are doing say development activities. The very first question that has come what need to be the spacing between these two fans? Now, let me put one point here, say we have realized the advantage of contra rotating fan and that is a reason rather going with a single fan or say maybe stage, we have gone with the contra rotating configuration.

So, these are the two contra rotating fans they have been ducted, okay. Now, what need to be the spacing between these two, because that's what will be deciding, suppose say it is a roadable vehicle, then we are having constraints with say sizing. And, that's the reason why we have explored the possibility for say what need be the spacing between these two rotors. And, we found that need to be say two times the diameter of the casing.

The benefit of contra-rotating we have explored with say different requirements say high rotational speed of rotor 1, higher rotational speed of rotor 2, both the rotors are rotating at design configuration. Basically, that's what will be helping us in order to manage the thrust. So, that's what we have done here. So, this is what is representing the total pressure contour at the exit plane. And, this is what is giving you new horizon for the say thought process for future development of flying taxis.

So, like, what all we are learning that's what is having say straightway application for the future development, okay. So, you can see here, earlier design that's what was high aspect ratio blade; here these blades are low aspect ratio blades and that too, that's what is handling different kind of flow configuration and their purpose, that's what is totally different.

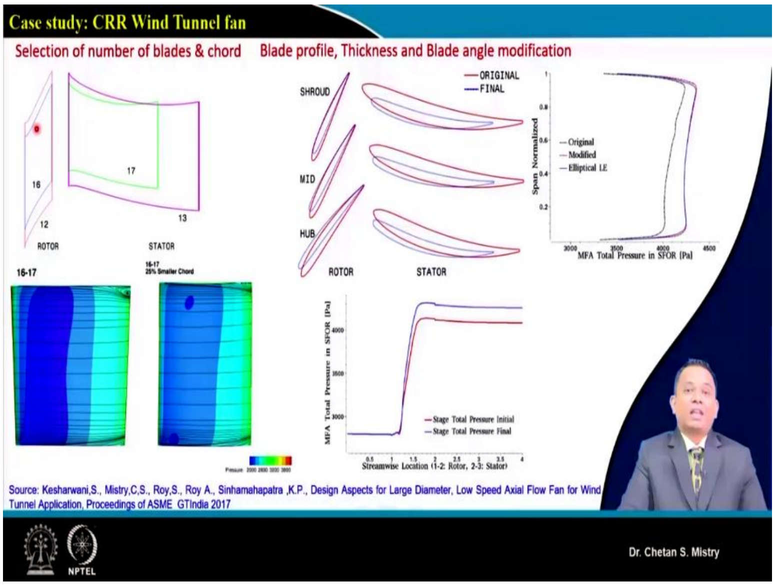
(Refer Slide Time: 20:27)



This is what is the case study what we have discussed in our week 11 for the design of say wind tunnel fans. So, this is what we already have discussed, say at IIT Kharagpur, Center for Railway Research. We are having say development of this closed loop wind tunnel; this project, it has been sponsored by RDSO, Indian railways. And, these are the requirements for the fans, that's what we have discussed, already discussed with say mass flow rate of 420 kg/s, the expected pressure rise in order to get rid of frictional losses, that's what is 1400 Pa, diameter is 4 m.

And, if you look at, this is how we have done our initial design for the fan, where we are having rotor and stator configuration. If you recall, we have discussed design in a different aspect, we have discussed with say design in terms of say having number of blades, we have discussed about say chord selection, we have discussed in sense of diffusion factor, then we have discussed about what all are the requirements of stator and rotor.

(Refer Slide Time: 21:48)



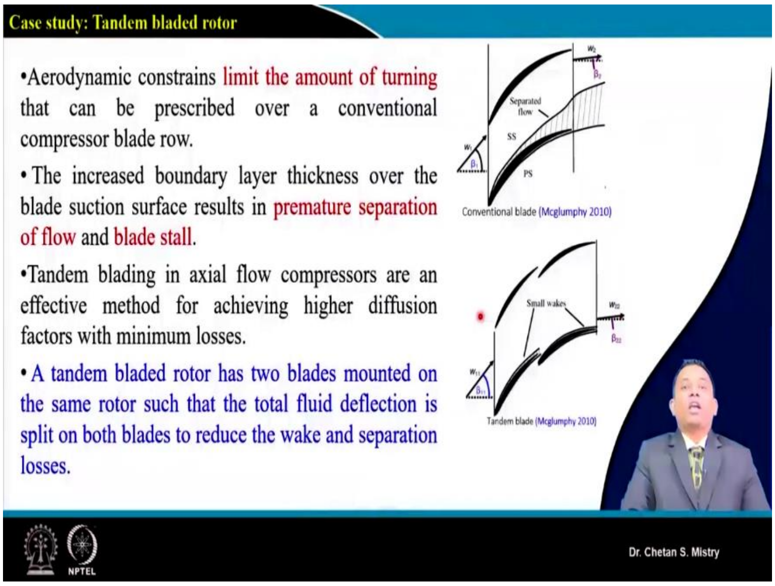
So, if we go ahead, this is what is representing. So, if you recall, this is what will give us idea with less number of blades and more number of blade for the rotor. We must realize, the passage between two blades, that's what is the defusing passage and that's what is following what all rules we have discussed in terms of length of the diffusing passage say diffusion angle, inlet area, outlet area.

So, based on number of iterations, we have come up with say combination of 16 and 17 blade. This is what is a design what we have discussed in week 11. And, we have explored our computational study and that's what is giving say best performance in terms of what we are expecting. So, here if you look at, say initial original blades were like this and later on we have done our modification in terms of number of blades, we have done our modification in sense of shape of the leading edge, we have done our modification in sense of chord length. And, if you look at, this is what is representing the final results or say finally, selected geometry.

Here in this case, this is what is representing say overall pressure rise or say pressure rise from the stage at the exit. So, when we were having our initial design, that time it was giving the pressure rise here, this is what is representing the modified design. And, here if you look at, this is what is representing how my stream wise pressure rise, that's what is happening.

So, here in this case, we can say, this is what is representing our stator and rotor combination, okay. And, this is what is our total pressure, that's what will be developed by this particular fan. So, as per our requirement, we need to do certain modifications. So, as we have discussed, we have done our initial design based on what understanding we are having. Later on, by using computational tools, modifications, they have been done to meet the requirement, okay.

(Refer Slide Time: 24:13)



Now, let us move to our next case study. So, we have discussed the present trend, it is looking for say compact and lightweight compressors. Compact in the sense, the length and diameter need to be smaller, lightweight in the sense number of components need to be lower. And, in order to achieve those requirements, per stage pressure rise requirement will be very high.

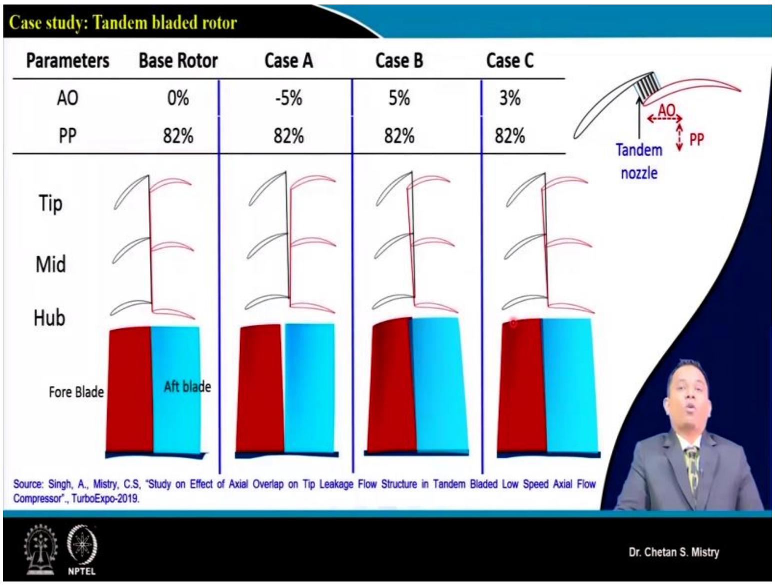
And, we have discussed in order to improve per stage pressure rise, we need to go with say larger diameter. Or maybe we need to go with say larger rotational speed or where we need to go with say higher axial velocity, we have discussed what all are the challenges, when we are talking about all these aspects.

Now, the situation is when we are having one more possibility is to increase the deflection angle. So, here if you look at, in order to achieve high pressure rise, my $\Delta\beta$ need to be larger.

So, if we are increasing our $\Delta\beta$; because of adverse pressure gradient, there are more chances for flow to get separated from the suction surface.

Now, when it is getting separated from the suction surface, we can say that particular airfoil or that blade, it is stalled. And very soon, whole stage will be going in surge and later on whole or all compressor stages will go in surge, that's what is very challenging aspect. So, in order to address this issue, people they have explored the possibility of tandem configuration where we are having two blades; they are being placed and considered as a single blade. So, in place of having one blade, we will be having two blades, that's what is been arranged on single rotor, I will show what exactly is the meaning of that.

(Refer Slide Time: 26:18)



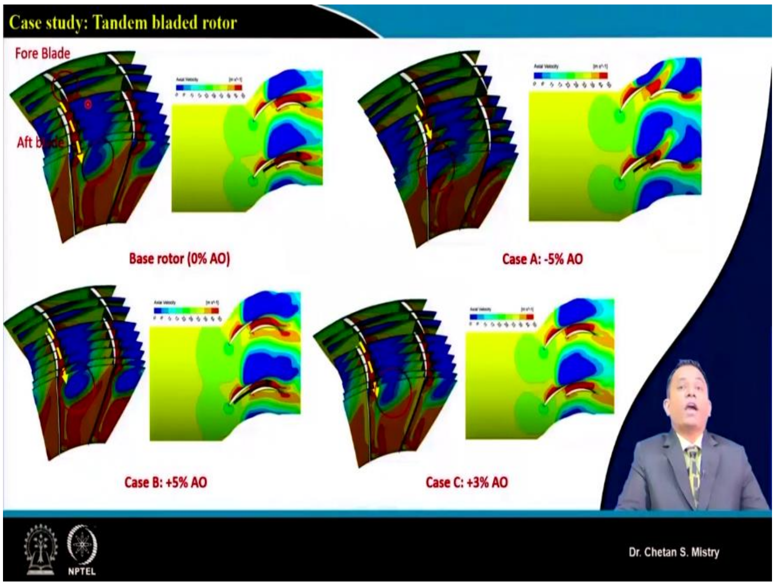
If you recall in week 3, we were discussing about what all are the possibilities for the design. Let me discuss here, what we learn for say tandem configuration, we want to take the benefit of flow. So, here this is what is the region, basically the trailing edge of my blade 1 and leading edge of my blade 2, that's what is acting like a nozzle and that is helping us in order to accelerate the flow on the suction surface of say next coming blade or blade 2. Now, when we are doing this, this is what is giving the benefit. This work, it has been explored initially for the wings and later on, people they tried to explore for say rotor as well as for stator also.

So, there are two main configuration we need to realize, one that's what is say my axial overlap. So, we can say this is what is representing the axial overlap and this is what is representing the percentage pitch. Limited work it has been reported in order to do the design for tandem bladed rotor because that's what is very challenging.

Now, number of data and results are available for cascades, but we can understand at particular station, all my configuration may be working fine. But after having background of all 11 weeks, you must realize, my blade, when we are talking, it is say for rotor it is not at particular station, this is what is three-dimensional entity. So, we need to think design in three-dimensional way.

So, this is what is represent 0 axial overlap, -5% axial overlap, $+5\%$ axial overlap, and $+3\%$ axial overlap. So, you can configure, near the hub axial overlap is 0 and at this location, near the tip region my axial overlap that's what is varying terms of percentage of say my chord.

(Refer Slide Time: 28:34)



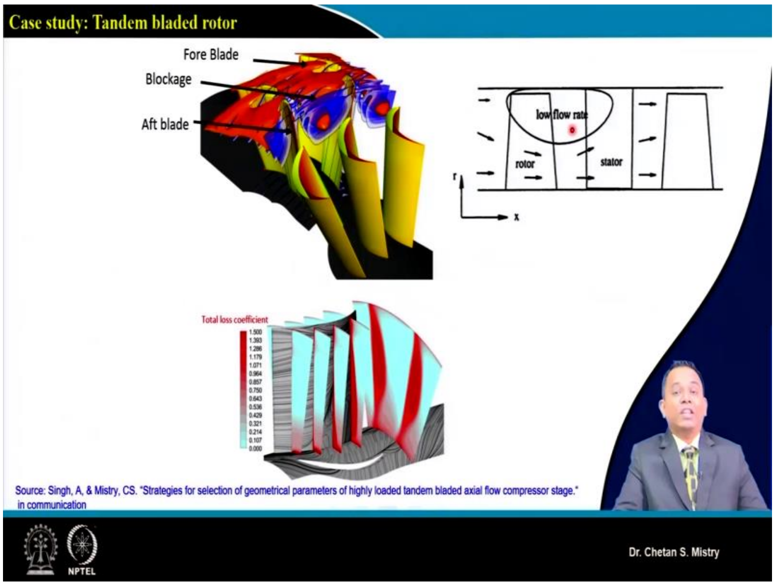
Now, these are some of the important observation as we have discussed, the use of computational tool that is very important. So, here if we look at, this is what is representing say the case for say 0% axial overlap. So, this is what is representing my axial velocity. If you look at carefully for the fore blade, we are having this as a tip leakage flow.

So, we need to provide certain amount of clearance between casing and the rotor blade. For our case, we kept our tip clearance as 3 mm. And, that's the reason, you can see this is what is a tip leakage flow, that's what is coming out from the fore blade. Now, this particular region because of this nozzle effect, we are having the acceleration of flow that is happening between these two blades.

But at the same time, the flow that's what is coming out in terms of tip leakage flow, that's what is say exiting here. At the same time, if you observe carefully, because of this turning of the blade we are having the separation that is happening near the trailing edge for our aft blade. Later on, this is what has been modified in terms of point say $5\%\ or -5\%$ axial overlap. Here, in this case, not much benefit in sense of nozzle effect it has been found and that is the reason why my trailing edge, say upper portion, say maybe 70% of the span, if you look at, there is a flow separation that is happening for aft blade.

So, this is what is not the promising configuration. Later on, it has been moved with the 5% and if we configure here, my acceleration of flow between this blade passage is such it is giving benefit in sense of removing the trailing edge separation, that's what was happening on aft blade. Later on, we have optimized with the 3% axial overlap and if we look at, my extent of this flow, what we say rollout or the secondary flow, what we can say the extent it is reduced, okay.

(Refer Slide Time: 30:58)

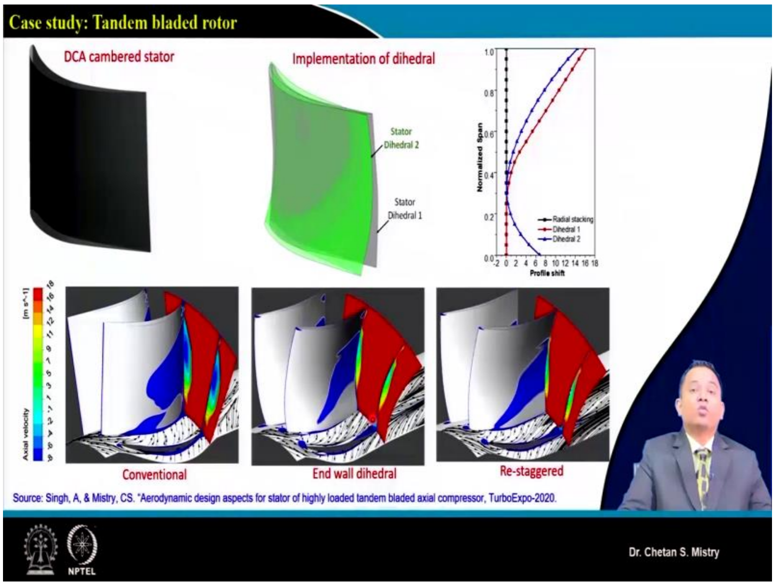


Now, if we look at, we have understood; suppose if we consider, say this is what is my rotor and stator combination, because of my growth of boundary layer along the casing, and tip leakage flow particular region, that is where I will be having low momentum fluid or say low mass flow rate. And, that's what will be acting like a pocket or say, 0 velocity kind of region, when that's what to be striking on the stator, the challenge for the design of the stator will be increasing.

So, here if you look at, this is what is a flow or the blockage, that's what is coming out from say, our tandem bladed rotor. Now, this is what will be striking on the stator. So that's what will be making the design for this stator to be very challenging. And, if you look at, this is what is say the exit total pressure loss coefficient. If we observe my exit total pressure loss coefficient, that's what is coming to be higher. We can say, we are having the separation of the flow, that's what is happening on the suction surface of the stator.

Let me put one important point here, say conventional stator and rotor combination only one way, that's what will be coming out from my rotor blade. And, that's what will be striking on the stator. Now, for this tandem bladed configuration, there are two wakes; one, from fore blade, one from aft, they both are striking on the stator. So, that's what is making design of this stator to be very challenging.

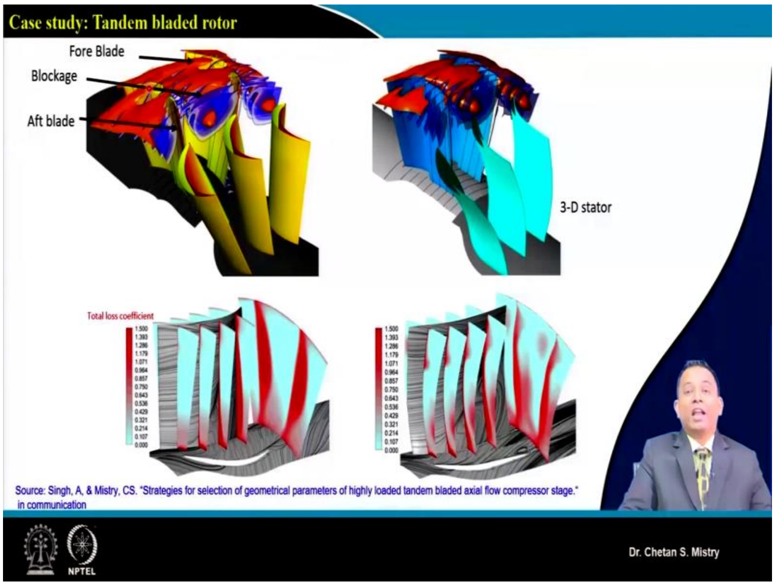
(Refer Slide Time: 32:40)



Now, these are the modifications this what we have done in order to achieve what we are looking for in terms of total pressure rise. So, this is what was initially been designed or conventional stator where we are having, say losses, that's what is happening. Now, later on it has been modified with end wall dihedral. So, you can understand, this is what is a three-dimensional stator.

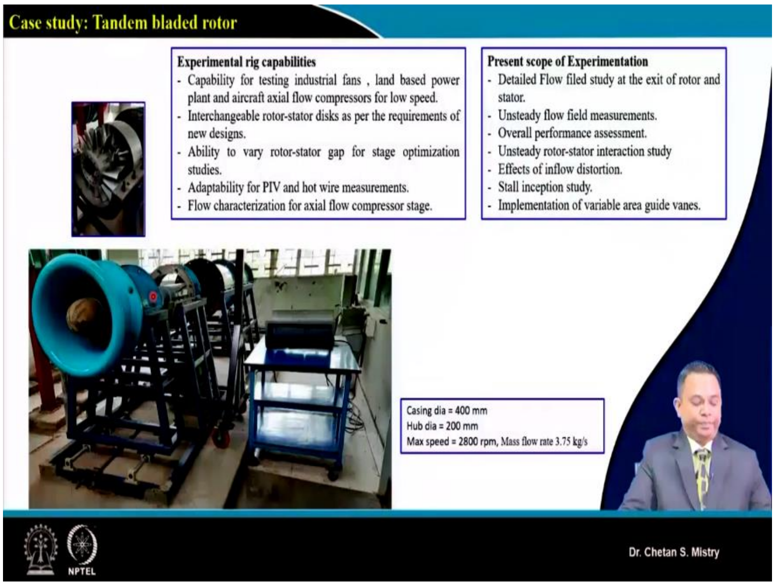
And, our computational tool, that's what is helping us in order to modify the design. For our quick solution, you can understand, like if I will be making number of iterations for all iteration if I will be making blade, and then after I will be doing my experimentation, that's what is required a lot of time. So, we are able to reduce number of iterations required for the experimental study. And, that's what where the cost is coming into the picture. Now, this is what has been final design with say Re-staggering of the stator blade. So, here we are able to manage our flow at the exit of the stage.

(Refer Slide Time: 33:57)



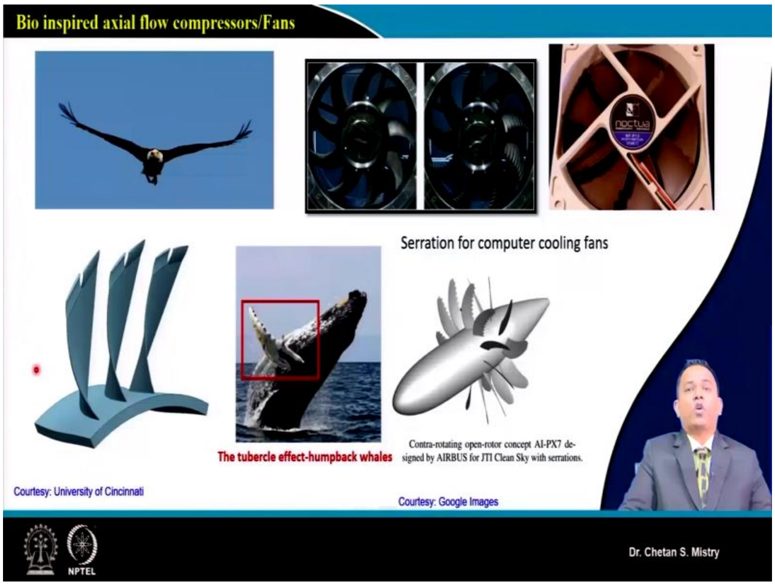
Now, this is what is representing, how my total loss coefficient, that's what is getting modified for both the stator configurations. So, this is what is a final state or final design of the stator for tandem bladed rotor. So, must realize, it is easy to say like using tandem bladed rotor, we can get high total pressure rise, but at the same time, the design for down coming stator will be very challenging. Design of rotor itself is challenging at the same time, design of stator also is equally challenging.

(Refer Slide Time: 34:42)



Now, this is what is experimental facility which has been developed at IIT Kharagpur. If you look at, these are our rotor blades and stator; and very soon, we will be having say outcome from the experimental results which will be the helping hand for the future researchers, as well as for the future development and help to the society.

(Refer Slide Time: 35:08)

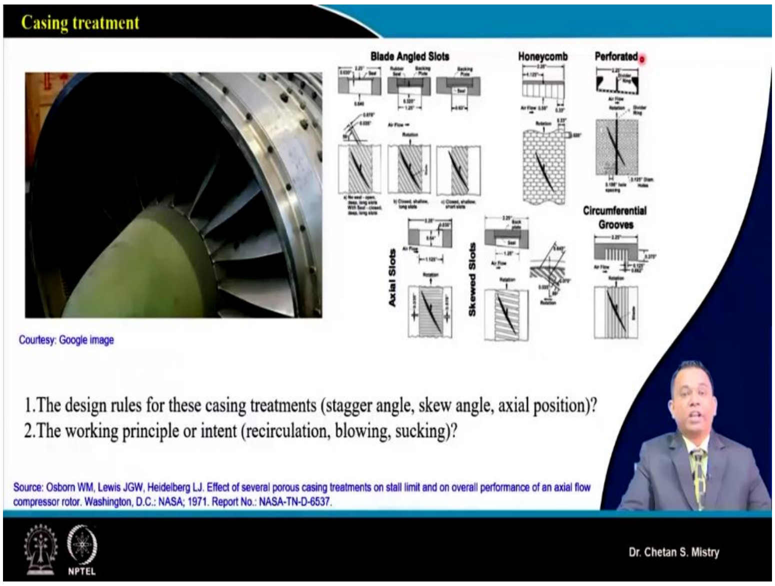


There are some more configuration what people they are exploring, that’s what is based on bio inspired, say compressor and fan blades. So, here if you look at, for this eagle, the end, say we are having tip region where these feathers are arranged, that’s what is helping them in order to minimize the drag. The same logic, that’s what has been applied at University of Cincinnati for the development of say compressor rotor blade.

So, they have splitted, this blade in two halves. And, by doing so, they are able to improve the performance in terms of operating range. Basically, how we are managing our flow in the tip region, that’s what is very important. Similarly, this is what is a concept called tubercle effect, based on say, humpback whales. So here, if we look at, that’s what is having some surrogated surface or that’s what is in technical language people used to say tubercles. This is what is helping whales in order to say minimize the drag and to propel.

So, same logic, that’s what has been used for development of say open fan configuration; even these days, for say cooling of say computers, servers, high speed computers, we are looking for cooling fans, they are making lot of noise and in order to address this issue, that surrogation, people they have adopted near the trailing edge, even people they are exploring surrogation at the leading edge. So, this is what is one of the bioinspired kind of configuration.

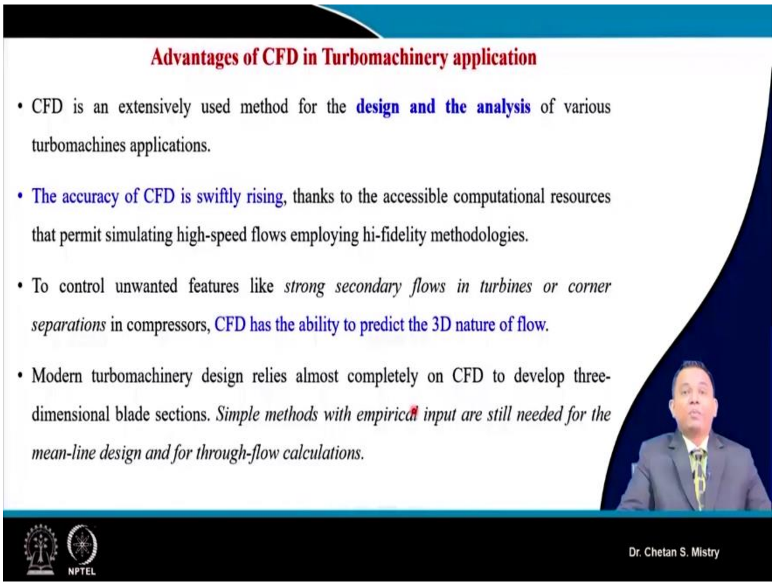
(Refer Slide Time: 37:02)



Now, people they are talking about, say casing treatment, that's what is to enhance the operating range and with having say certain compromise in the efficiency. So, there are different configuration, people they have explored. It is called say blade angle slot. So, here you can look at, this is what is a blade angle or blade airfoil at the tip region, some honeycomb section, perforated kind of configuration, circumferential groove, screws slotted or axis say slotter or say axial slots, people they have explored.

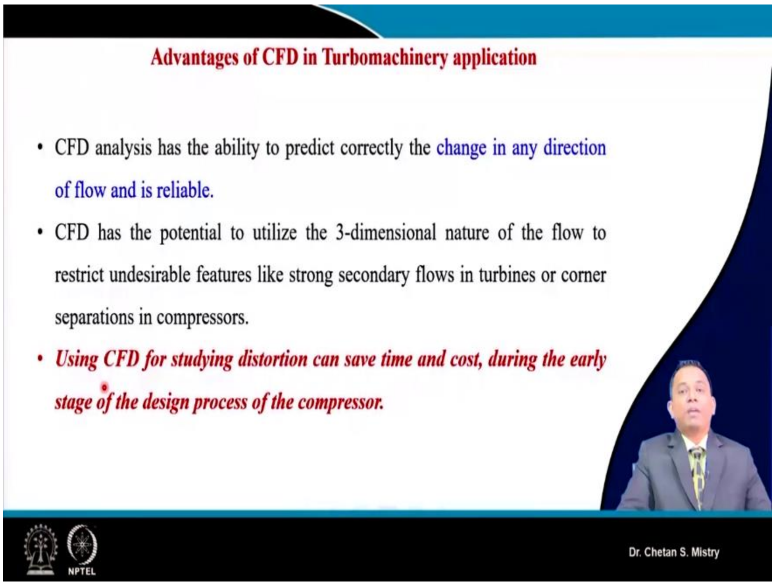
Now, in order to decide this kind of configuration, the computational tool, that's what will be helping a lot. There are many open literature available, where people, they have explored their computational tool to finalize the geometry for the casing treatment. Now, there are different configurations say stagger angle, skew angle, axial position, that's what is very important, maybe we need to go with say recirculation, maybe blowing, sucking all those terms, that's what will be coming into the picture and computational tool that's what will be helping hand for this kind of simulations.

(Refer Slide Time: 38:21)



So, in overall if you look at, CFD is now helping hand for the design and analysis for different turbo machinery. Now, the use, that’s what has been increasing day by day, because we are able to simulate high speed flow with high fidelity methodologies. Now, we are able to control say secondary flow phenomenon; even say, the corner separation for the compressor and turbine blade, and that’s what is helping us for analyzing the three-dimensional flow that is happening between the blade passages. Say, modern turbomachinery design, that’s what is relied completely on CFD, but still I am putting point here like CFD is analysis tool, it is not the design tool. So, initially, you need to do your preliminary design using your excel sheet what all we have discussed; later on, you can use this as a tool for the simulation and finalize your geometry.

(Refer Slide Time: 39:29)



Now, CFD, that's what is giving very good prediction in sense of change of direction and reliability of the results. This is also helping in terms of say three dimensionality, what we have discussed secondary flow, say tip leakage flow, say corner separations, those all terms that's what is now we are able to achieve. There is a recent trend, it says this CFD, it has been used to solve the problem of distortion.

So, for future engines, we are looking for say distortion tolerant fan that need to be used. And, at the design stage only if we are taking care of this is what will be helping us for the development activities. So, here we are stopping with, we have discussed about different case studies, what all we have explored over the year for the design and development activities.

Even at the same time in order to do number of iterations in design, we have discussed some of the bioinspired configuration, we have discussed about the casing treatment and we realized we are having the advantages of using this CFD analysis for axial flow compressors and fans. So, here we stop with, now in next lecture, we will be discussing about very important aspects, we say what all are the challenges of CFD application to axial flow fans and compressor. Thank you. Thank you very much.