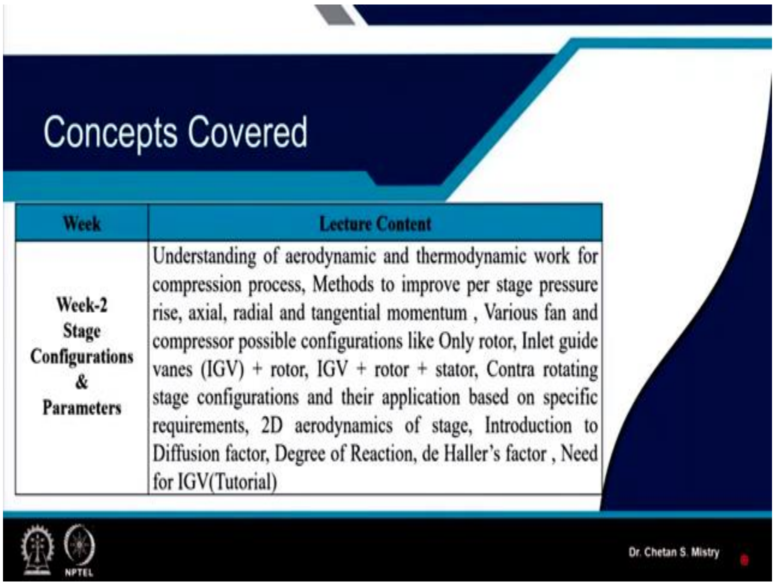
Aerodynamics Design of Axial Flow Compressors & Fans
Professor Chetankumar Sureshbhai Mistry
Department of Aerospace Engineering
Indian Institute of Technology, Kharagpur
Lecture 16
Design Concepts

(Refer Slide Time: 0:30)



Hello, and welcome to lecture-16. In our module 2, we were discussing about the detail understanding of aerodynamic and thermodynamic work for say compression process. We have discussed about different approaches in order to improve per stage pressure rise, we were discussing about say changing your pressure rise or improving your pressure rise by using increase of peripheral speed, increase of axial velocity, increase of deflection angle and we were discussing what all are the limitations for those concepts.

Now, people they are addressing those issues, in order to improve the per stage pressure rise considering that as opportunity. Then, we were discussing about what all are the different configurations which are possible for our axial flow compressors and fans. We have considered these special applications, where we are using single rotor or only rotor, with inlet guide vanes and rotor configuration.

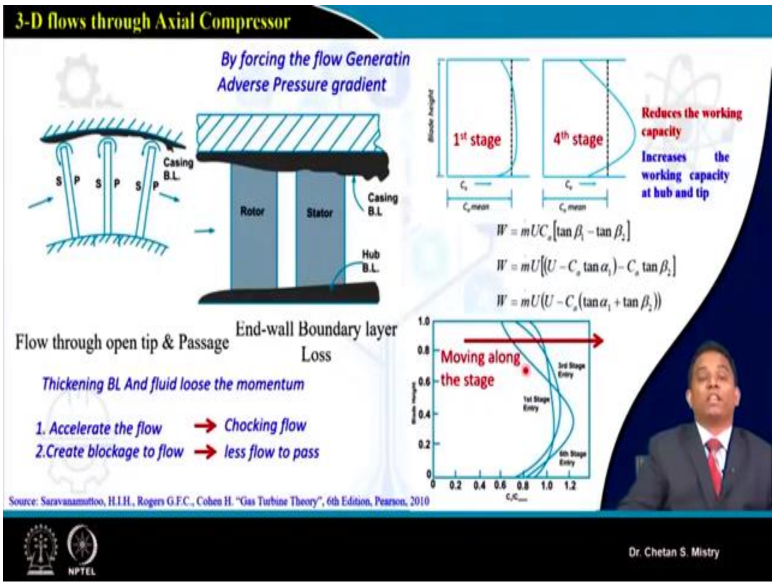
We have discussed about rotor and stator configuration, we have discussed about say inlet guide vanes, rotor and outlet guide vane configuration and as we have discussed that's what we say 1.5 stage configuration. Then, we were discussing about the most important and most on going research work, that's what was with contra rotating concept. So, that's what has given us idea how do we decide and what all are the benefits of using different kind of configurations.

Then we were discussing about 2D aerodynamics of our stage and we have introduced our parameters called diffusion factor. So, that’s what we have learned; in order to estimate the diffusion for per stage or for all your stages, during your calculation, you need to do the calculation for diffusion factor. This diffusion factor also help us in way we can count the number of blades. So, how many blades you will be using that also can be estimated if you are assuming your diffusion factor to be some number. And we have seen that number as per the recommendation that’s what is say maximum 0.5.

Then, we have introduced the parameter for degree of reaction and that degree of reaction is equally important parameter that’s what is giving us idea, whether we will be having our diffusion that’s what is happening in our rotor or only in stator or both in rotor and stator. So, based on that we will be doing our calculation for design; or degree of reaction, we have discussed, that’s what is thermodynamic measure of a diffusion process.

Then, in last session we were discussing about the need for inlet guide vanes, we also have discussed about the application of tandem configuration for axial flow compressors and fans. We have seen, these days people, they are doing their ongoing research for development of tandem rotor as well as tandem stator configurations. With this, now, let us move ahead; we will be moving more towards say design aspects. So, with this all fundamentals we need to add on something that’s what will lead to help us in sense of designing the axial flow compressors and fans.

(Refer Slide Time: 4:36)



So, let us try to understand what is a three-dimensional flow or how you will be having your three dimensional flow in axial flow compressor. So, what all parameters we were discussing about diffusion factor, about degree of reaction, those all are the two dimensional parameters. Even if you recall, the velocity triangles also we are discussing in two dimensional geometry.

Now, let us try to look at here, say… this is what we can say, it is a casing and here we are having our hub. So, when we are having our rotor and stator combination, we know it is working under adverse pressure gradient and that is the reason if you consider, if my casing and hub their solid body, you will be having your growth of boundary layer that will be happening along this wall.

This will continue if we are considering we are having multi-stage configuration that will continue till the last stage of your compressor. So, you can understand, if you are having growth of boundary layer at the first stage that will be having the boundary layer thickness to be lower and maybe it may possible the last stage, the whole your blade height will be covered with the growth of boundary layer from both hub as well as from the casing side.

And we can understand, if you are considering this growth of boundary layer, that's what is acting like a low momentum fluid and that is nothing but it is also acting like a solid body. So, we can understand, our flow passage, because of growth of boundary layer from both hub side as well as from say your casing side, it will be making nozzle kind of shape, okay.

And if we are considering, we are having say… diffusion process, that's what is happening along with that we are having this decreasing area that's what is giving you acceleration of flow and this together that's what is making your flow to be more complex within the flow passage, okay. And this acceleration of flow because of nozzle action that's what will lead to accelerate your flow and that's what is putting your limit to chocking flow, okay.

Now, if we consider say for rotor and stator combination, we know our rotor, that's what is a rotating component and for rotor to rotate we need to provide certain amount of clearance between casing and rotor. So, here we can see, we are having clearance that's what is provided. Now, as per our understanding of fluid mechanics and aerodynamics, we are having the flow movement from high pressure side to low pressure side.

So, if we consider, say this is what is my blade, I am having one side that's what is say pressure side and on other side I say it is a suction side. So, I will be having my flow that will be moving from pressure side to the suction side, because we are providing certain amount of clearance. So, this is what is a natural process, that's what is bound to happen.

Now, you can understand, we are having growth of boundary layer, that's what is happening from casing wall and along with that we are having the flow that's what is we are defining as tip leakage flow that is coming from pressure side to the suction side. And that's what is making whole my flow to be very complex and we can say, this is what is giving a three-dimensionality to my flow within the flow passage, okay.

So, because of presence of this growth of boundary layer between casing and the hub and because of my tip leakage flow, if you look at here, this is what is representing my blade height versus what is happening with my axial velocity. So, if you look at what we are assuming? We are assuming our axial velocity to be constant. So, when we are doing our design for axial flow compressor, we are assuming our axial velocity to be constant.

And because of this growth of boundary layer and because of presence of this leakage flow, we will be getting the axial velocity profile like this. So, near the hub and near the casing region we will be having the decrease in axial velocity. And as per our continuity, somewhere that need to be increased. So, here if you look at, say mid-section or near the mid span you will be having slight rise in axial velocity.

So, this is what is happening at the first stage. Now, let me put what is happening at say my 4^{th} stage? So, if you look at, this is what is very interesting! You can see, I will be having, say… my profile that's what is got distorted near this casing region as well as near the hub region. I suppose to get my axial velocity like this; in place of that my axial velocity profile that has changed, okay. Now, what is happening?

In this particular region, we are having our axial velocity to be lower, near this casing I am having my axial velocity to be lower and somewhere in the mid span, I will be having my axial velocity to be higher. Now, what we know from our fundamentals? We can say, my work done, that's what is given by this relation.

$$W = \dot{m} U C_a [\tan\beta_1 - \tan\beta_2]$$

Now, based on my understanding for velocity triangle, I can replace this $\tan\beta_1$ in sense of U and $\tan\alpha_1$, okay.

$$W = \dot{m}UC_a[(U - C_a \tan\alpha_1) - \tan\beta_2]$$

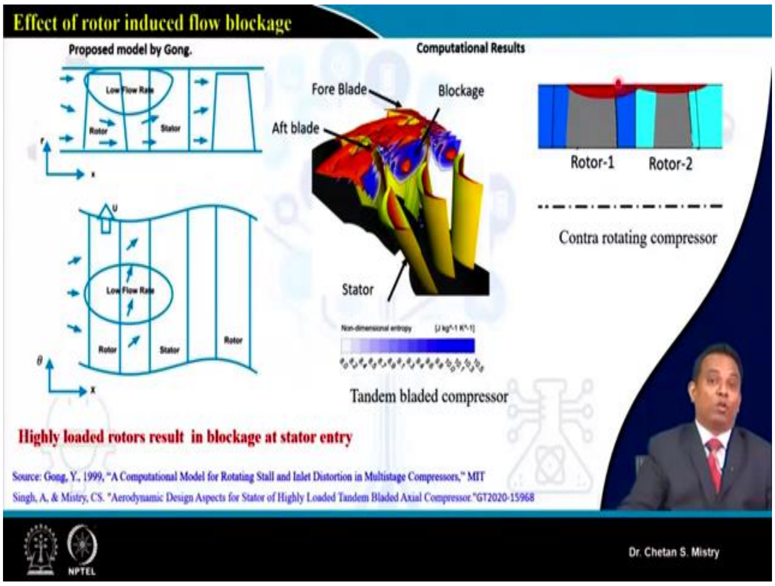
For initial design consideration, if we consider, say my α_1 is constant and I am considering suppose it by β_2 also is constant; if that's what is your configuration, you can say, you know in this particular equation we are having $U - C_a$ So, we can say this nomenclature, this bracket form, that's what is say constant. If you are considering here, near the hub region and near the casing region, what is happening? Our axial velocity is reducing.

So, what it means? That will be increasing my working capacity near the hub and tip, this is what is from our understanding of this equation. Now, what happens near the mid span, we are having say increase of my axial velocity that means that's what will lead to reduction in my working capacity or pressure rising capacity.

So, in overall sense if you are looking at, we have detrimental effect of growth of boundary layer or the three-dimensionality of the flow, that's what is happening because of my growth of boundary layer and my tip leakage flow that lead to three-dimensionality to my flow. And, that's what will be reducing my pressure rising capacity of particular stage.

So, here if you look at, we are having say… the profile for the first stage, you can see, this is what is a distortion in my profile for axial velocity and as I am moving far, say for the sixth stage, if you look at, my whole velocity profile that's what got distorted. So, you can understand, based on our assumption for constant axial velocity, we are not achieving what axial velocity we are looking for.

(Refer Slide Time: 13:09)



So, now the question is, if this is what is your case, what need to be done, okay? So, let us try to understand what exactly is happening within my flow process. So, this is what is a model proposed by Gong. Say, if you look at, this is what is my rotor that will be followed by my stator and what we have learned? Because we are having say… rotor that's what is having the clearance here.

So, because of growth of boundary layer and because of presence of your tip leakage flow, I will be having low flow rate or low momentum fluid, that's what will be present near this, you know, upper span of my blade. So, this flow, that's what will be when striking on my stator, my stator it is been designed with the assumption of particular angle means we are assuming our axial velocity to be constant. In place of that because of presence of this low momentum fluid, my flow incident angle to stator will change and that's what will lead to deteriorate the performance of my stator.

And in overall sense you can say, for what it is been designed it will not give me what pressure rise I am expected from that particular stage, okay. Suppose if I consider, I will be having say… longer length blade, say… we can say high aspect ratio blade, we will be having the presence of low momentum fluid in between the span maybe around say 50% to 70% span.

So, this is what is creating more challenge in sense when we are talking about highly loaded rotor. We will be having the blockage of flow that's what will be happening in my flow structure. So, if you look at here, this is what is our work, that's what has been reported; if you look at, this is my tandem bladed rotor configuration, it is very interesting! Here in this case, we are considering tip leakage flow that's what is coming only from single rotor.

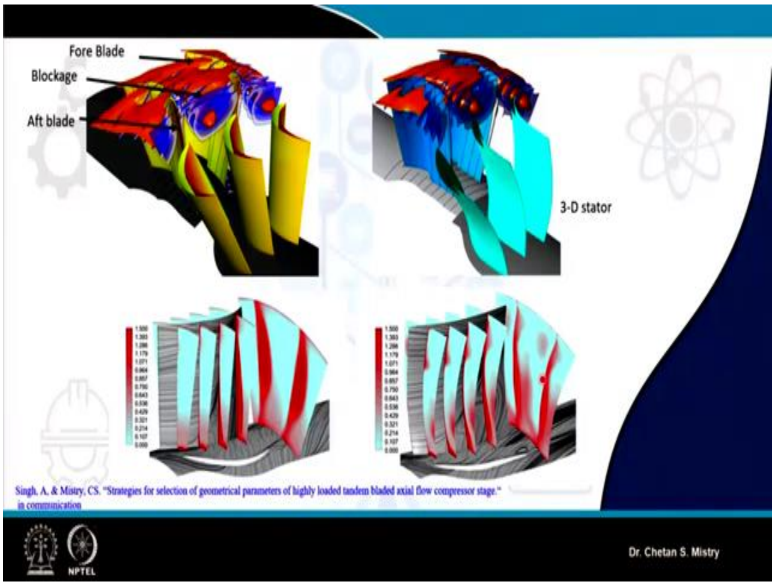
Here in this case if you look at, my tip leakage flow, that's what is coming from two rotors. So, this is what is my rotor one and this is my rotor two, you can say that flow leakage that's what is happening twice. So, because of presence of this twice tip leakage flow, you can understand, here, I will be having my low momentum fluid that's what will be coming out or we can say, it is a vortex that's what will be coming out it will be striking on my stator and that's what will be making my design for this stator to be very challenging, okay.

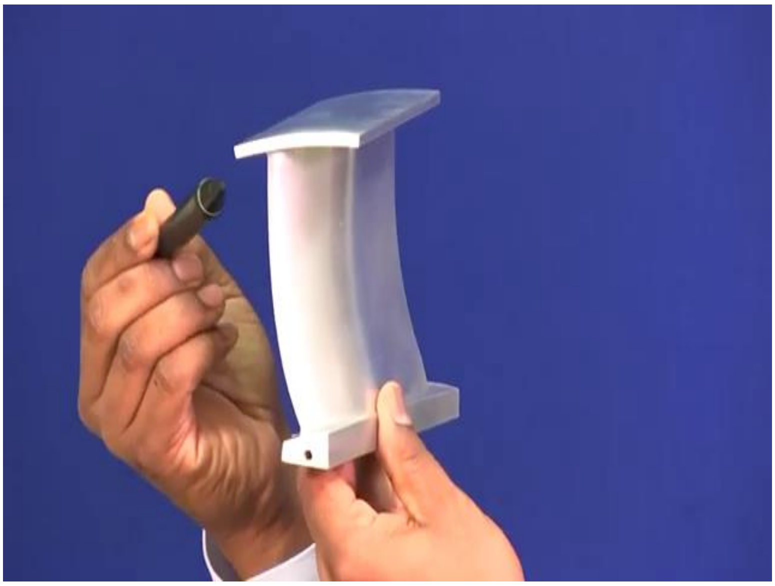
Now, one other work, that's what is going on at IIT Kharagpur; that's what is for the development of Contra rotating fan configuration. So, here if you look at, this rotor, it is rotating part and that's what will be followed by the stator, it is a stationary part. In the case of Contra rotating fan, we know one rotor, that's what is rotating in clockwise direction and my other rotor that will be rotating in counter clockwise direction.

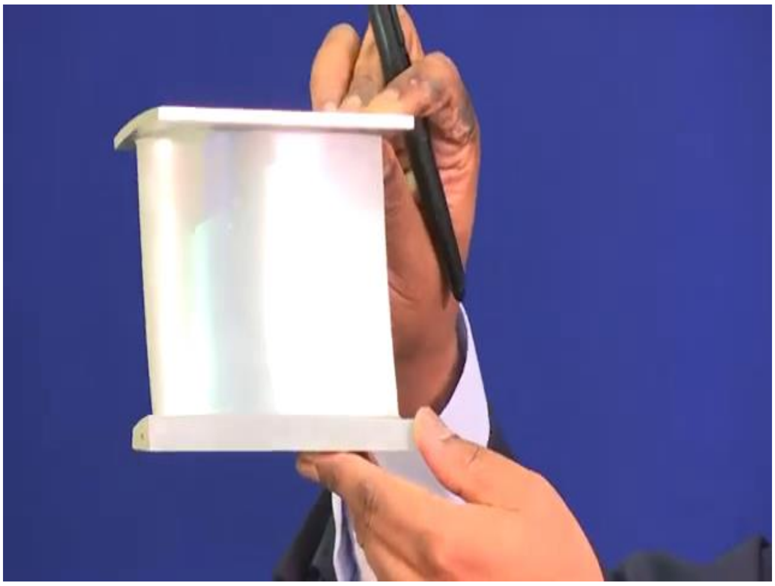
So, let us see what is happening there? If you look at here, suppose say, if this is what is a flow structure, that's what will be coming out from my rotor 1. Now, for this case what was happening? My flow that will be incident on my stator and that is the reason why, you know; since, this stator is a stationary part, I will not be having further movement of my fluid. But, when we are talking about the contra rotating concept, you know, I will be having suction of this flow that's what will be happening for rotor 2.

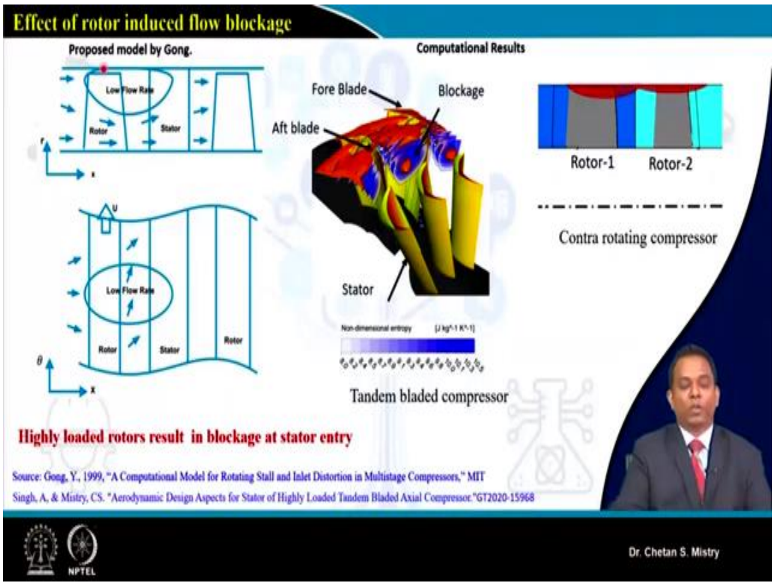
And if you look at, I am able to manage my flow, I am able to minimize the effect, that's what has happened because of growth of boundary layer and because of growth of your tip leakage flow. So, you know, according to our understanding, we need to move forward with. So, this is also one of the advantages for using contra rotating concept. So, as and when required we will be discussing about what all are the benefits of different kind of configurations.

(Refer Slide Time: 17:46)









This is what is our on going work on tandem bladed rotor. So, if we look at carefully as I mentioned earlier, we are having two blades, fore blade and Aft blade. So, for rotor as we have discussed in our earlier case, we have our tip leakage flow that's what is coming from pressure side to the suction side for the single blade. Here in this case, we are having tip leakage flow, which is happening for both fore blade as well as for aft blade.

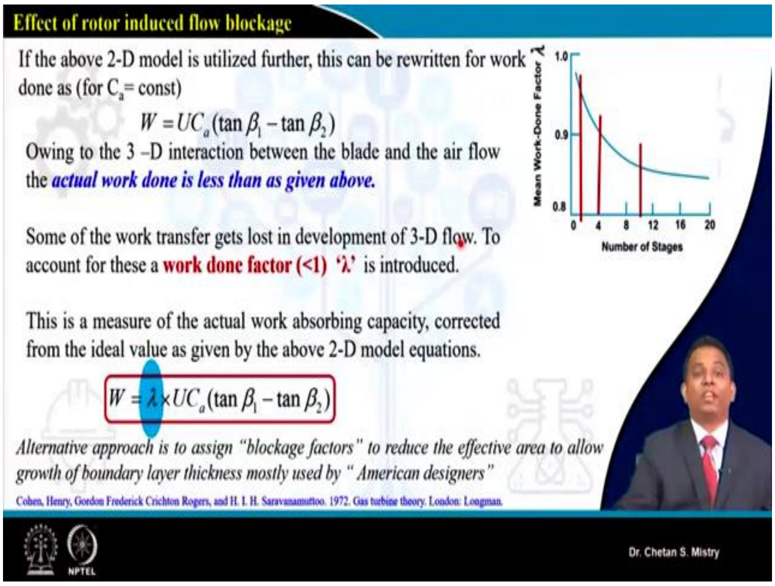
So, in sense of flow structure which is coming out from this tandem rotor, we can say, it is making the blockage to our flow. Now, this blockage that will be striking on the next coming stator. So, in this particular region, where we are having the striking of this blockage flow, that's what will lead my stator to work under off design condition. And, that is the reason why, if you look at, my flow that's what is coming out from the exit of the stator, it will be having say different kind of flow structure.

Now, since we have designed this stage for particular design point it may possible that will be working fine under design condition. But when we are talking about off design condition, it may lead to flow separation or it may lead to have say stall of stator. In order to address this issue, we have modified our design, that's what lead to this kind of stator. That stator we can say, it is a three-dimensional stator.

Now, the stator, if you look at carefully, at the exit, we are able to manage our flow. If we look at carefully, so this is what is the stator which has been designed the downside of my tandem bladed rotary. Here if you look at, this is what is my leading edge and my whole blade it is been stacked about the leading edge and this is having a three-dimensional shape to this blade. One interesting case, because we are having our tandem bladed rotor, which is having say high deflection angle, so from that high deflection angle to exit to be axial, we need to have a specific design of the stator.

And, if we look at the stator, that's what is having higher cord length, which will be taking care of what we saying in terms of say development of my flow on this stator surface, okay. So, here if you look at, this is what is my leading age and trailing age; and that's what will lead to give us the performance what we are expecting in sense of our stage. So, experimental work, that's what is going on in order to have the performance assessment of this tandem bladed stage.

(Refer Slide Time: 21:13)



Now, what we have realized, it is say what we have calculated using our fundamentals of two-dimensional equation, that's what it says my work done it is given by

$$W = UC_a(\tan\beta_1 - \tan\beta_2)$$

Because of presence of your three-dimensionality in the flow, my actual work done that will be less than that of what we are calculating, okay.

So, some of the work transfer that gets lost because of development of three-dimensional flow and that's what has been taken care by introducing the parameter, that's what is called 'work done factor'. So, here this is what is my work done factor (λ), it says my work done factor, that's what is less than 1.

So, let me modify my equation. So, work done, that's what we are calculating now by using

$$W = \lambda \times UC_a(\tan\beta_1 - \tan\beta_2)$$

Do not get confused with this terminology of work done factor, it has nothing to do with your efficiency. Basically, when we are doing our design that time we need to take care of my actual flow condition and in order to consider that actual flow condition at the initial stage only we need to introduce our work done factor, okay.

So, this is what is a reported work by Cohen and Rogers and according to them say… there are some numbers that's what is given that's what is representing my work done factor and on x axis we are having number of stages. So, suppose if I am considering my first stage design, I will be selecting my work done factor in the range of 0.96, 0.97. Suppose say, if I am designing my fourth stage, I will be selecting my work done factor in the range of 0.9 and if I will be moving ahead, I will be having my work done factor that's what will be in the range of 0.86 or 0.85.

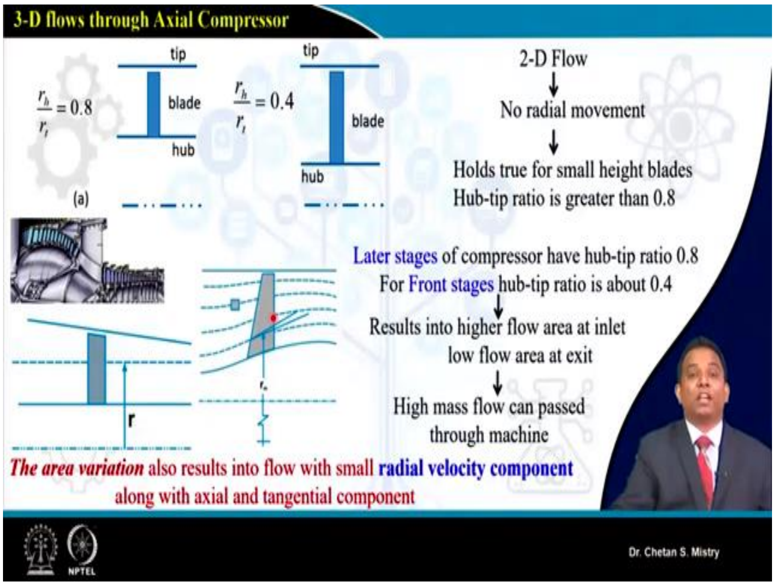
So, you can understand, say as a precautionary measure for doing our design, we need to take care of the three-dimensionality of the flow by incorporating the factor it is called work done factor, okay. And now onwards when we will be discussing about the work done we will be introducing this parameter called say your work done factor, okay.

Now, there are different approaches, that's what is adopted globally. Mainly different engine manufacturing companies, based on their experience, they have done a whole lot of computational study, experimental study, theoretical study and based on that they have developed their own factors, okay. And those factors they are introducing when they are doing their design.

One of the concept that's what has been adopted by Americans, that's what is called your blockage factor. This blockage factor that has been calculated based on your momentum thickness, that's what is based on your boundary layer thickness. So, which stage you are designing, you need to be careful, do not forget to incorporate your work done factor, okay.

So, now onwards we will be discussing all in sense of our work done factor, it says this is what is taking care of our flow three-dimensionality. So, what all flow three-dimensionality we are discussing, that's what will not end here.

(Refer Slide Time: 25:05)



Let us see what all we know in sense of our blades. So, here if you look at, this is what is representing my blade, okay. If you consider the centre lines for both the blades are same, one that's what is representing my hub to tip ratio as 0.8; second, that's what is representing my hub to tip ratio as say 0.4.

So, here if you look at, for what I am having my hub to tip ratio as 0.8; you can say, this is what is shorter blade and here if you look at, this is what is my taller blade. Do not get confused with the terminology of aspect ratio, your aspect ratio is parameter which is correlating the height of the blade and chord of the blade and this is what is hub to tip ratio.

So, you can consider when we were discussing our initial part, say in last session, in last module, we were discussing everything in sense of two dimensional flow, where we have not considered any radial movement. It holds true when we are having smaller height of blade, which is having hub to tip ratio in the order of say maybe 0.8 or say more.

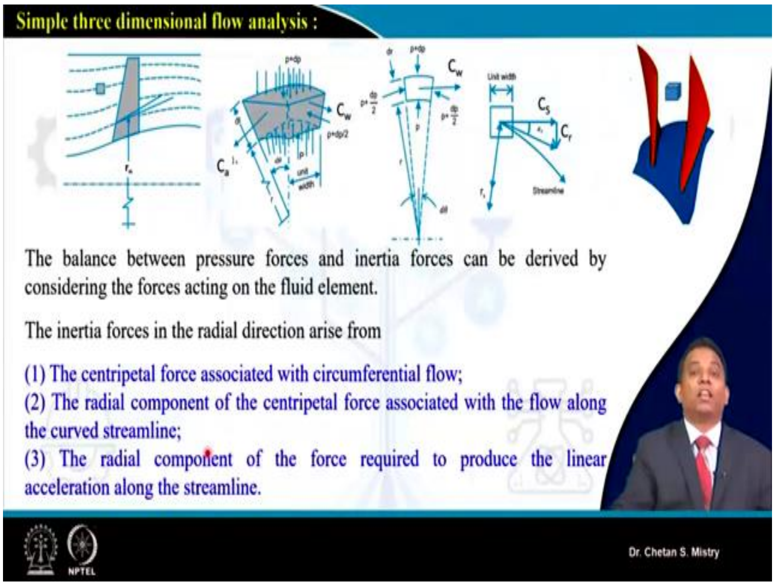
So, for later stages, we are having the ratio of hub to tip, that's what is in the range of 0.8 and for front stages, we say for fan or maybe for LP compressor, this ratio, it is in the range of 0.4. So, this number that may be vary when we are talking about land based power plant. So, this is what is indicated number, it has nothing to do in sense of design at this moment this is what is giving rough numbers, okay.

Now, what is happening? We will be having because of this the change of my height of the blade we will be having our flow passage that's what is coming to be a different shape. So, here if you look at, this is what is representing constant hub diameter kind of configuration, where my tip diameter that's what is changing. And here if you look at, we have discussed, say this is what is in line to what we were discussing about our fan. So, for fan, my inlet area and my outlet area that's what is changing.

Now, if we consider I will be having say constant area at the entry and the exit, we can say our stream lines that will be parallel stream lines. And what we have realized? Because, we are having the change of height and that change of height is nothing, that's what is because of change of my density. And according to my continuity my exit area will be coming to be lower. So, what my stream lines will be doing?

Say for casing region my stream lines that will try to follow the casing wall near hub also it will try to follow the hub curvature and that's what will be giving the three-dimensionality to my flow. And because of that, we will see, we will be having one more velocity component that is nothing but radial velocity component along with your axial and tangential velocity what we were discussing earlier. So, now you can say, the three-dimensionality of the flow that is happening because of what flow track you are having, okay.

(Refer Slide Time: 28:45)



Now, if you try to look at, say suppose if I consider, this is what is my blade; so, for this blade, I can take my element within this flow passage. Just remember here, this is what is representing, it is for the representation purpose only. So, if I consider this is what is my particle, that's what is between these two blades; I can take that elemental part, that's what is having say radius r and elemental radius or say that distance as dr.

I can say this angle as a $d\theta$. If I will be putting my pressure, I can say my pressure, that's what is acting from downside as P and from upper side I can say $p + dp$, on the other side I can say $p + dp$. Now, if you look at carefully, this is what is a direction, that's what is representing my flow direction, it is axial velocity and if you consider this is what is my circumferential direction, that's what is representing my whirl component, okay.

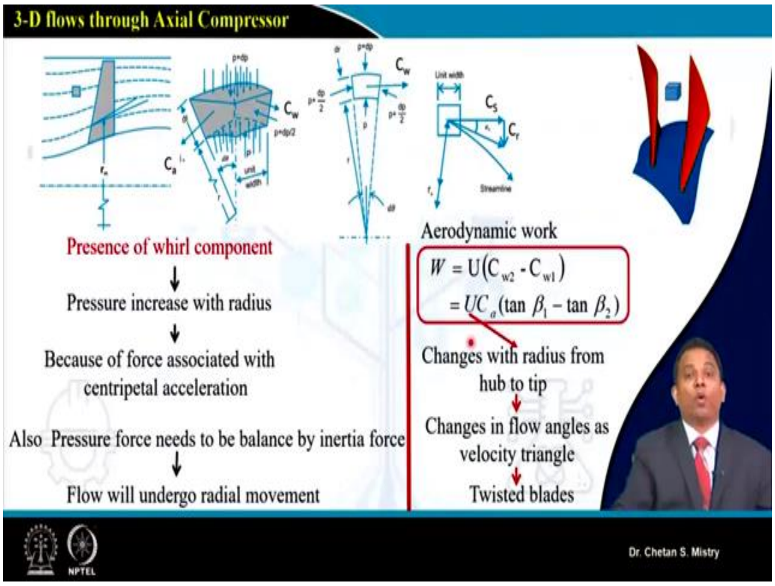
Now, if I will be taking this as a space, you can see, I will be having my whirl component, that's what is acting in this direction and I will be having my pressure forces acting on this element. On other way if I am looking at, you will see, we will be having the streamline flow that's what is happening within this passage and I will be having one more parameter, I say, it is a radial velocity component, okay.

Now, what we need to understand here is what all flow that's what is happening, it is because of balance of my forces. So, here if you consider, we are having two forces which are acting on this element; one, that's what is my pressure force and second, that's what is by inertia force. And now we are coming with the term, that's what is the inertia force acting on this element. So, from where this inertia force is coming?

It says, the centripetal force associated with the circumferential flow, you can say, this is what is a centripetal force, that's what is acting because of my circumferential flow; radial component of centripetal force associated with the flow on curved stream line, you can say, this is what is representing that force component, we are having relative velocity component coming into the picture.

And we are having radial force component, that's what will be producing the linear acceleration along my stream line. So, you can understand, we are having two different kind of configuration; one, it is saying my pressure force and second, that's what is talking about my inertia force. So, now you can understand between the passage, we are having the presence of inertia force and pressure force. So, we are moving more towards the detailed understanding of flow what is happening within our passage, okay.

(Refer Slide Time: 31:56)



So, now in sense of our understanding for the three-dimensionality of flow, we can say, we are having the presence of whirl component and that's what is increase; say, it says, that's what lead to increase in pressure along my radius and this rise of pressure is because of force that is associated with my centripetal acceleration and also this pressure force that need to be balanced by my inertia force that will be giving my fluid particle a radial moment, okay.

So, now because of presence of this inertial force, because of change of my radius, we are having the change in the movement of my particle. So, our flow is going more three dimensional within the passage, okay. Now, from fundamental what we learn? Say our axial, say compressor, that's what is generating the aerodynamic work. In fundamentals we are writing it is

$$W = UC_a(\tan\beta_1 - \beta_2)$$

So, this peripheral speed, that's what is a function of my radius and you can understand, since this is what is a function of my radius, so, U that will be changing from hub to tip and that's what will be giving my velocity triangle with different angles and that's what will lead to highly twisted blade. So, now in overall if we try to understand, our flow is no more two-dimensional flow, our flow configuration, that's what is highly three-dimensional.

And this three-dimensionality, that's what is happening because of presence of my growth of boundary layer, because of presence of my tip leakage flow, this is what is because of what kind of flow track we are having, this is what is because of what is happening in sense of dynamics on the fluid particle within the blade passage and also because of three-dimensionality of my blade that's what is highly three-dimensional blade.

So, with this, we are making our initial understanding of three-dimensional flow through axial flow compressor. In next session, we will be discussing how we will be using this concept for development of different design aspects. Thank you, thank you very much for your attention!