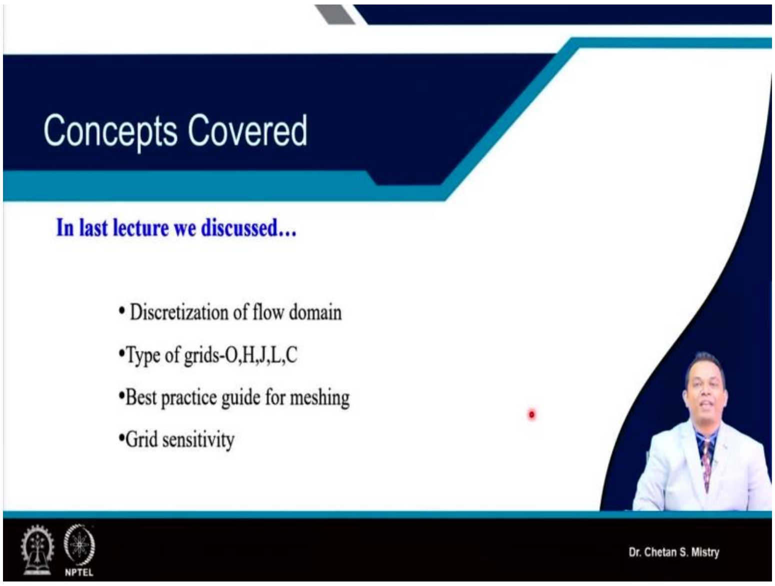
Aerodynamic Design of Axial Flow Compressors & Fans
Professor Chetankumar Sureshbhai Mistry
Department of Aerospace Engineering
Indian Institute of Technology Kharagpur
Lecture 67
CFD Application to Design and Performance Assessment (Contd.)

Hello, and welcome all to lecture 67. We are discussing about the CFD application to design and performance assessment.

(Refer Slide Time: 00:38)



We have started our discussion with discretization of the flow domain. We are discussing about the CFD analysis that's what we are using for analyzing axial flow compressor as well as for axial flow fan. We know discretization it is art basically, we need to understand what exactly we are looking for in terms of analysis.

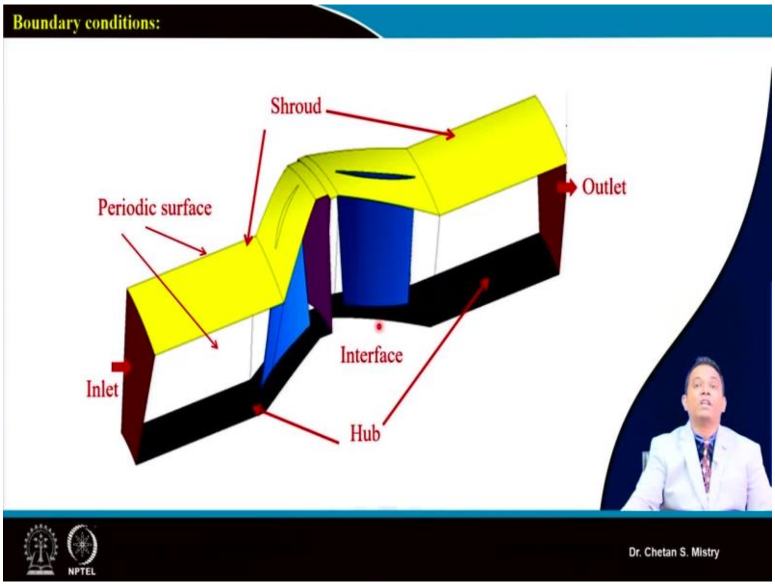
There are different kinds of grids we have discuss, say O-grid, H-grid, J-grid, L-grid, C-grid, they are having special applications. So, this is what is the special kind of configuration what we are using, because analysis…CFD analysis of turbomachinery, CFD analysis of axial flow compressor is a specialized analysis where we are looking for so many in detail study, where we need to select this element in order to capture specialized flow physics.

We also have discussed about the best practice guideline for the meshing. Where we have discussed about what kind of analysis we are doing based on that how many number of elements we need to go with? Even we were discussing about the y^+ value, y^+ basically we are calculating what need to be say the height of my layer next to my blade. And, we have discussed it is preferred to have y^+ value to be below 1 or nearly 1, that's what is art basically, in terms of doing say meshing.

So, when we are doing meshing, that time, we need to be careful about all these aspects. Then we were discussing about say grid sensitivity or we can say grid independent study. There are different approaches; different researchers, they are following, different companies they are following and based on that they are representing their detail flow field based on mesh sensitivity.

Now, after this once we have done our meshing, next requirement, that's what is to have the solution to the problem or we want to solve the flow domain now. In order to solve that flow domain, we are looking for the term that's what is called different boundary conditions.

(Refer Slide Time: 03:23)



So, let us try to understand what exactly is the meaning of boundary condition? So, here in this case, we can say suppose say this is what is say stator and rotor combination. Here in this case, if you are looking for, say very entry point, that's what is called inlet. Now here, my domain what we are making, that's what needs to be thoroughly understood.

Suppose if I consider this is what is say my rotor blade, so from leading edge, I need to put my inlet domain say 1 chord or 1.5 chord upstream so that my flow that's what I am putting here that will be going inside my rotor blade, and that's what will be giving us what we are looking for, okay. Many times, based on the application where we are using this fan, people they are considering the inlet domain. So, this is what is my inlet domain.

Now, here in this case, same way we are having outlet domain, this is what is my surface we can say this is what is representing my outlet configuration. So, in open literature, most of the time Best Practice Guideline it says my outlet domain need to be 2 chord downstream of from my trailing edge of say rotor or stator or at the exit of my stage.

So, from here 2 chord downstream we will be putting our outlet flow domain. So, that's what we will be giving sufficient amount of flow domain for my solution to get converse. If you are putting too short, it may be possible that it will not give what analysis or what we want to do analysis for. So, we need to be very careful when we are analyzing say single stage this is what is a configuration.

Suppose say we are analyzing multi stage then we must be having numbers in terms of what is a gap between two stages. So, that gap we will be putting here and for exit we will be configuring with the outlet domain in the same way. Once we have decided with this, say here in this case, this is what is say my blade surface, this is what is representing the hub. Hub in the sense the disc on which or drum on which we are fixing our rotor blade as well as stator blade, okay.

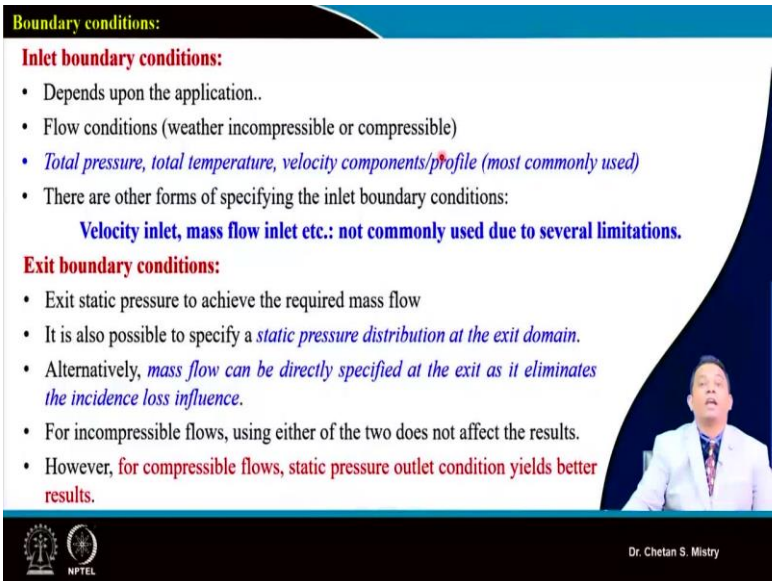
So, here in this case, this is what is representing say this dimension what we can say as a hub. Same way, on top side this is what is my casing we can say that as a shroud. Now, this shroud that's what is representing my say inner diameter of the casing, okay. Now, kind of stator and rotor what we are using accordingly we need to fix that one maybe hub surface or say on shroud surface.

Let me tell you there are different kind of configuration, that's what is possible with the stator. Mainly, people they are understanding this stator that's what is acting like a supporting structure, but many compressors, industrial compressor as well as say aero engine compressor in which the stators are also of cantilever kind in which that will be having gap between say hub and blade surface. So, under that condition, we need to provide certain amount of gap here.

Now, this is what is say my one domain we can say it is say my rotor domain, this is what is say my stator domain. Now, in order to analyze the stage, we need to put certain kind of interface here. So, this is what is representing the interface. So, this is what is my rotating configuration, this is what is my say stationary configuration and in between that's what I say interface.

And, on both the sides if you look at, this is what is representing the periodic surfaces. So, this is what is my periodic surface 1, this is my periodic surface 2. So, we will be discussing what we mean by these boundary conditions today.

(Refer Slide Time: 07:47)



So, let us see. Suppose if we consider say inlet boundary condition; so, in order to start solution for our analysis, we need to give certain boundary conditions, okay. It is like you are forcing your computer to do what you are looking for. That means it will not understand what exactly you want to analyze, you need to say this is what is my inlet and in inlet we need to put certain say flow parameter.

So, basically what kind of application we have based on that my inlet boundary condition we need to give. So, flow conditions basically is it say compressible flow or incompressible flow, you know better now, when we are considering say subsonic compressor, when we are configuring say transonic compressor, that's what is differentiating in sense of compressible flow and incompressible flow.

So, total pressure, total temperature, velocity component or profile, that's what is most widely been used as the entry boundary condition or inlet boundary condition. There are other form of boundary conditions also possible. Say, we can put say velocity inlet, mass flow inlet, but these boundary conditions have their own limitations.

Now, suppose say we are having our experimental data available with us; in sense, we have done our measurement of inlet total pressure by using say our pitot tube, okay; or we can say, we have used our probes, okay, multi rake probe, with that we will be having distribution of inlet pressure. So, that profile; suppose say, we have used multi hole probe in order to measure the inlet velocities before my say rotor or stator. So, that velocity profile also can be taken as inlet boundary condition.

We will discuss when I will be talking about the case study that time, we will see what is the effect of taking those actual experimental results also.

Now, when we say we have given inlet condition, at the same time, we need to say this is what we are looking for at the outlet, that’s what is called say exit boundary condition. So, this exit boundary condition, people they have preferred to go with say static pressure outlet, in order to achieve required mass flow rate,

Just understand one thing, basically, we have discretized our domain; so, each element what we have selected, that’s what will be solving our three equations, continuity equation, momentum equation and energy equation. Now, by solving that, we are getting different parameters like pressure, velocity, temperature, density, all those calculations that’s what it is doing. So, must realize this is what we can take as one of the boundary condition.

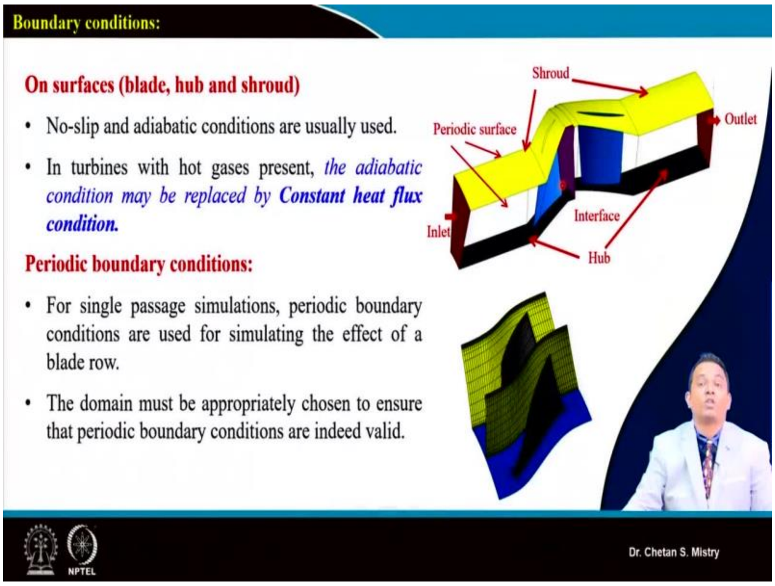
So, sometimes people, they are putting say static pressure distribution at the exit that is also can be considered as one of the boundary condition. Many times, mass flow outlet condition also people they are preferring with, but it says it is not capturing the flow incident configuration. So, when we are looking for specific kind of analysis under that configuration, this boundary condition will not help.

Now, for incompressible configuration, both the conditions are valid with either you give say static outlet condition or static pressure outlet or atmospheric outlet condition and or say maybe mass flow rate outlet condition.

So, for compressible flow analysis, when we are analyzing say transonic fan or say transonic compressor, under that configuration, we need to put static pressure as a boundary condition, okay. And that too, you need to put in an interval because just understand your computer is not understanding your code will get set up off calculation of different parameters and that’s only will be helpful to you in order to capture the socks.

So, there is a technique for that in which at certain interval after a few iterations, people they are changing the static pressure at the outlet and that is how they are getting what we are looking for in terms of solution to the problem.

(Refer Slide Time: 12:25)



Now, once we have given inlet boundary condition, outlet boundary condition; now, what we know the surfaces what we have, say hub surface, shroud surface, and blade surface, these all, they are solid walls. So, here in that case, you need to take say no slip boundary condition; you no better what is the meaning of no slip and free slip condition, okay.

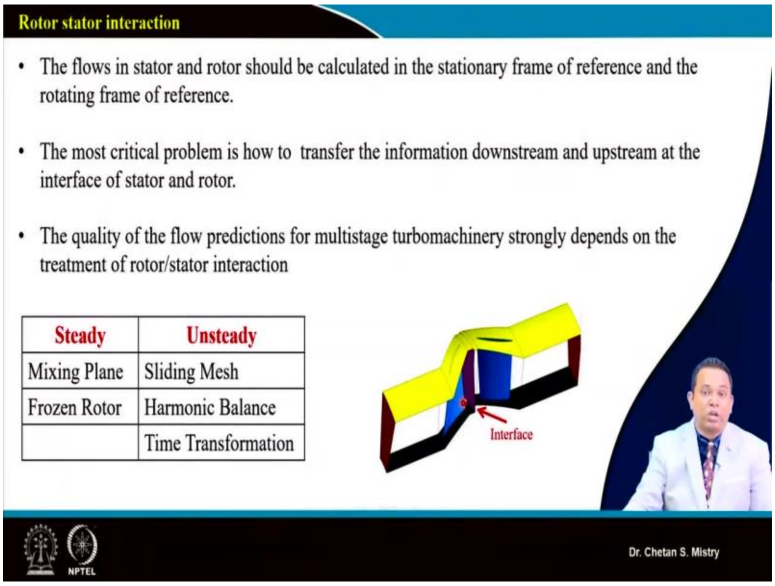
Now, when we are looking for growth of boundary layer on say hub surface, that's what is affecting my flow field, same way we have understood the growth of boundary layer from the casing, that's what is changing my flow field at the exit of rotor or stator that is also equally important. We have realized the suction surface is very sensitive in terms of growth of boundary layer, because we are having adverse pressure gradient.

So, in order to capture all those details, we need to go with say no slip condition with adiabatic wall. Now, many times when we are analyzing say heat transfer study, under that configuration in place of say adiabatic wall we need to go with say constant heat flux kind of configuration.

Now, next, that's what is your periodic surface. So, here as we have discussed, this is what is representing my periodic surface 1, this is what is my periodic surface 1, that's what is used to exchange the data between the blade passage. So, this is what is used for simulating the effect of blade row. So, we need to select this condition of periodic boundary condition or say we need to select that surface or we need to select or we need to make this periodic surface such that it will exchange the data between these two rotor blades.

So, basically in order to save your computational power, in order to save say time, we are analyzing our flow domain by using this periodic flow configuration. We are analyzing maybe 1 blade or 2 blades; it depends what kind of flow physics we are looking for.

(Refer Slide Time: 14:44)



Next, that's what is the exchange of data between rotor and stator. So, just understand here, we can say when we are having say stator say supposed this is what is my stator, this is what is my rotor. So, we can say, rotor that's what is working under rotating reference frame, here this is what is in stationary reference frame.

So, now we need to exchange our data. So, here whatever is happening in terms of flow domain for rotor that data needs to be transferred to the stator domain. We know we cannot make single domain in which we will be putting rotor as well as stator because this is what is say working under rotating configuration and this is what is working under stationary configuration that's what is very challenging aspect for analysis; those who are dealing with CFD analysis of turbo machinery of stage configuration, this is what is very challenging.

So, in order to analyze that, we are putting the interface here. So, this is what is a region that's what is called say interface. So, there are different kinds of configuration which are available which people they have explored over the years; for steady analysis, it says mixing plane approach or they say frozen rotor approach. For unsteady analysis, they are adopting with say sliding mesh, harmonic balance, time transformation, different-different methods, that people they are using with. So, the selection of interface that's what is very important for us.

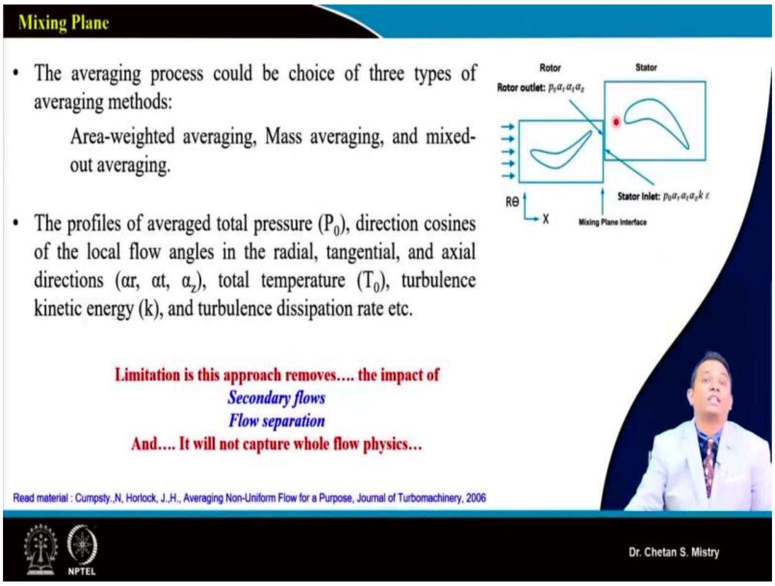
(Refer Slide Time: 16:24)

So, let us try to look what we mean by mixing plane approach. So, the thing is that this is what is a simplest way of analyzing say interface between rotor and stator, what it does? Say, it assumes the exiting flow from stator or rotor, suppose I have inlet guide vane before my rotor or when maybe I will be having rotor. So, what it will be doing? It says like when it is reaching to this interface, the flow will become say uniform and that uniform flow, that's what will be entering downstream of rotor stator domain.

Now, what it is doing? It is doing pitch wise averaging of flow solution, that's what is transforming from one domain to another domain. So, basically the flow data what we have at the mixing plane, that's what is getting averaged out in a circumferential direction. So, suppose say, the flow that's what is coming out from suppose say rotor. So, when it will be reaching to say interface region, the flow parameter they are getting averaged out.

So, circumferentially average out this data. Now, that data that's what will be transfer to the stator domain. Now, this is what is very tricky, and this is what is very challenging aspect when we are looking for analyzing the stage, okay.

(Refer Slide Time: 17:57)

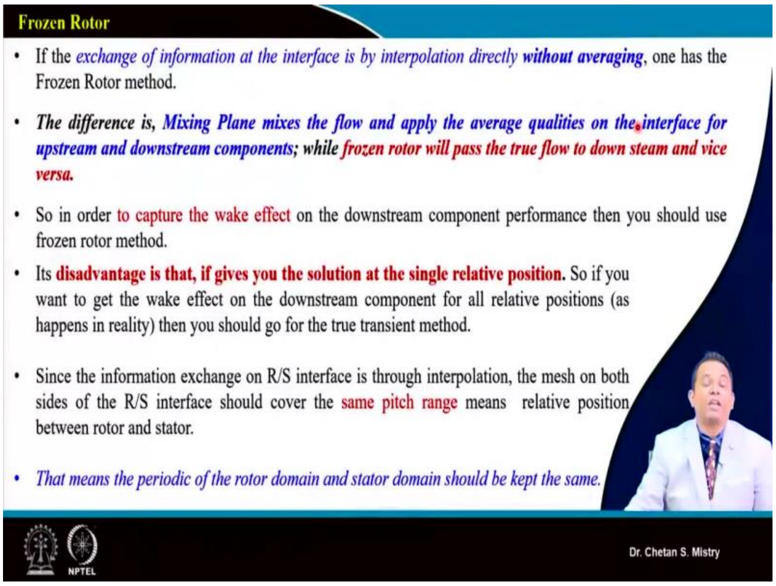


Now, suppose when we are doing this averaging, we are having different averaging techniques that's what is available. One, that's what is mass flow averaging, area averaging or next that's what is called say mix out averaging. So, here in this case, if you are looking at, say this is what is my rotor outlet. So, at this plane, we will be calculating all our parameter those are circumferentially averaged parameter.

Now, those parameters that's what will be the input for my stator domain. There is nothing wrong in analyzing this data. This approach it was been proposed by Denton in 1980, when we were not having that maturity and understanding in terms of computation, computational course as well as say computational facilities.

Now, here in this case, what will happen when we are exchanging our data, the data exchange that will not give what we are looking for in terms of understanding of secondary flows or flow separation, even we are unable to capture the wake, that's what is coming out from the rotor. So, wake that will be coming out from the rotor it will get averaged out at the interface and when it is going to the next domain, it is average value. So, exact flow field detail that cannot be captured here, okay.

(Refer Slide Time: 19:29)

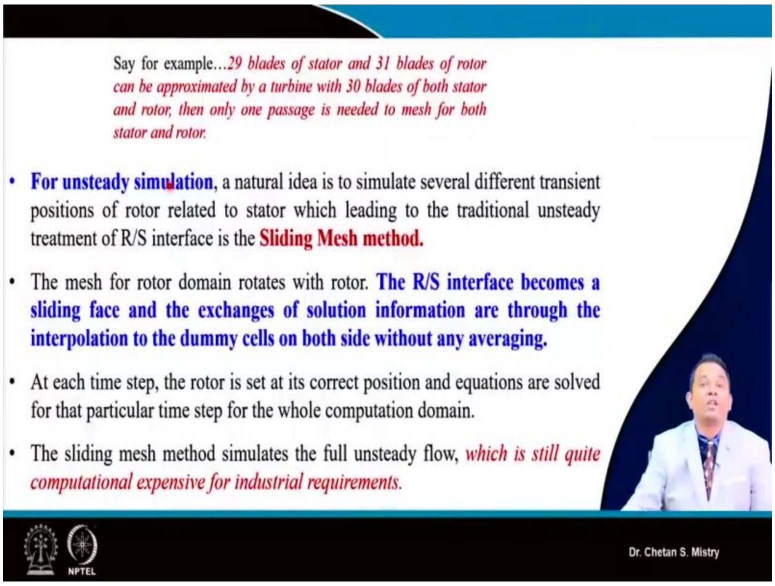


Now, in order to capture this detail, we are looking for this kind of flow, where we can capture our flow separation, where we can capture our secondary flow, where we can capture our wake formation, then we need to go with the different approach. That approach it is called frozen rotor approach. So, what it is doing? Say, this mixing plane, that's what was doing averaging at one plane, and that's what has been transferred to the next domain. While for say frozen rotor, it is transferring the data straightway, okay.

So, whatever data that's what has been captured at the exit of my rotor plane, the same data, that's what will be transferred to the stator. Now, the here challenge is in sense of the parameter called pitch ratio. So, pitch ratio selection that's what is very important here, what we mean by pitch is suppose say how many numbers of blades we are having. So, the relative position between rotor and stator, that's what will be changing the flow field.

Suppose, if I consider I'm having say, even number of blades for rotor, I will be having say odd number of blades for the stator. Under that configuration, we need to set or overlap our domain in the interface region such that the exact exchange of data, that's what will be happening there.

(Refer Slide Time: 21:00)



Suppose if we consider, this is what is one of the example. Say, suppose we are having 29 number of blades for the stator and 31 number of blades for the rotor. So, we can assume both the blades to be say 30. And, based on that we will be doing say setting up this flow domain such that the exchange of data, that's what will be happening in a proper way.

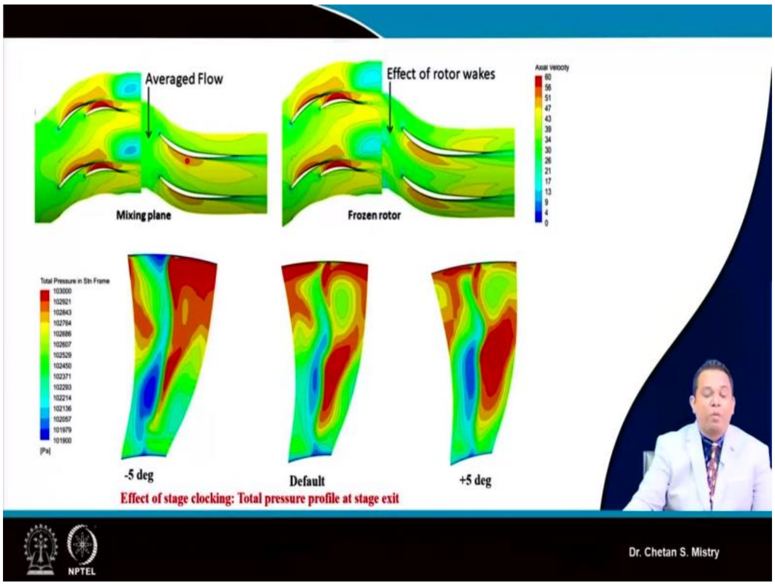
So, when we will be discussing our case studies, that time we will be discussing about how my results, that's what is changing when we are taking this frozen rotor approach. Now, with this next approach, that's what will be coming is say unsteady simulation. So, for this unsteady simulation, mainly what we are looking for it is the exchange of data between rotor and stator, that's what is happening with a sliding mesh approach.

Now, under that configuration, whatever data we are analyzing at the exit of say rotor domain, by using say dummy cells, it will be doing all calculation that data that's what will be transferred to say next coming say stator blade. And, this is what will be helping us in order to understand detail flow field.

Now, when we say we are looking for this kind of configuration, at the time, we are looking for say high computational power. Now, when we are talking about the specialized requirements, suppose say, we have done our initial design, after that we are looking for say detail flow field analysis for that only we can go with this kind of configuration else it will be very expensive affair.

So, mainly when we are doing our analysis, initial stage or design stage analysis, we are going with say mixing plain approach or frozen rotor approach and based on that we will be finalizing our design stage. Then later on in order to understand detail flow field, maybe we will go with say unsteady simulations.

(Refer Slide Time: 23:07)



Now, here this is what is representing say what we have discussed in terms of mixing plane approach and frozen rotor approach. So, this is what is the simulation results from one of my PhD student. Here if you look at my flow that will be coming out with two wakes here. We can say, this is what is near this tip region kind of configuration. Under that condition, what will happen? Say, my flow, that's what is getting averaged out.

So, here if you look at the wake, that's what will be coming out, that's what is getting averaged out near the interface plane, and that average value that's what is transferred to say next coming stator. So, when we are using this kind of approach, we are unable to understand how my wake that's what will be impacting on downstream stator, okay. And, that study that's what is very important for us, okay.

Now, in order to analyze this kind of configuration, here if you look at, the same analysis, that's what has been done by using frozen rotor approach. So, here we can clearly see what wake that will be coming out that's what will not be getting averaged out, but that's what is transferred to downstream stator.

Now, what we were discussing in sense of putting say pitch or say clocking effect or how we will be setting our…so, our stator downstream. So, here if you look at, we are having slight discrepancy in terms of positioning, this is what can be set by having say clocking effects study.

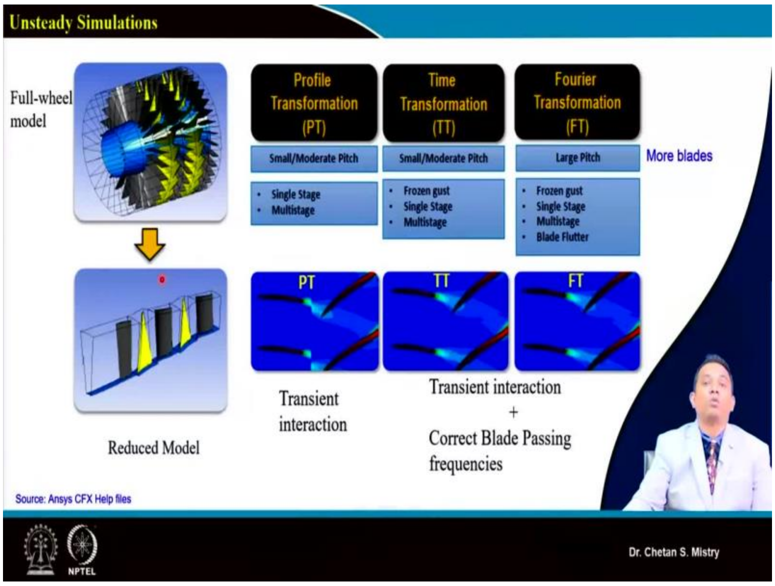
So, when we are using say approach that's what is called frozen rotor approach. It is always advisable to go with say clocking effects study or pitch effects study.

So, here if you look at, this is what is coming at the exit of the stator. So, here this is what is by default this kind of configuration what we have selected and this is what is representing or my flow that will be coming out from the stator. When we are shifting our say stator domain in say $-5° \, and + 5°$, you can say left hand side and right hand side, you can do both way maybe rotor position also you can change or maybe you can change stator position, relative position we need to change.

So, here if we look at, when we are putting this kind of configuration, we will be having this particular low momentum fluid, that's what is coming out from the stator it is large in the size. Here, this is what is a normal configuration, here this is what is. So, we can say, exact flow field study, that's what can be captured by using this kind of configuration. So, we need to be very careful when we are selecting this approach; say frozen rotor approach and mixing plane approach.

So, as and when you are doing analysis, we need to be careful what approach we will be selecting with, what is our interest! Basically, all thing that's what is coming is what we want to analyze, based on that we need to use different approaches.

(Refer Slide Time: 26:26)



Now, in order to do unsteady simulation; so, now the computational power that has increased, people they are becoming rich in terms of having computational facility in universities, even for industries and that's a reason why they are analyzing say unsteady kind of flow. So, there are different approach that's what has been defined by ANSYS CFX; they say Profile transformation, Time transformation, and Fourier transformation.

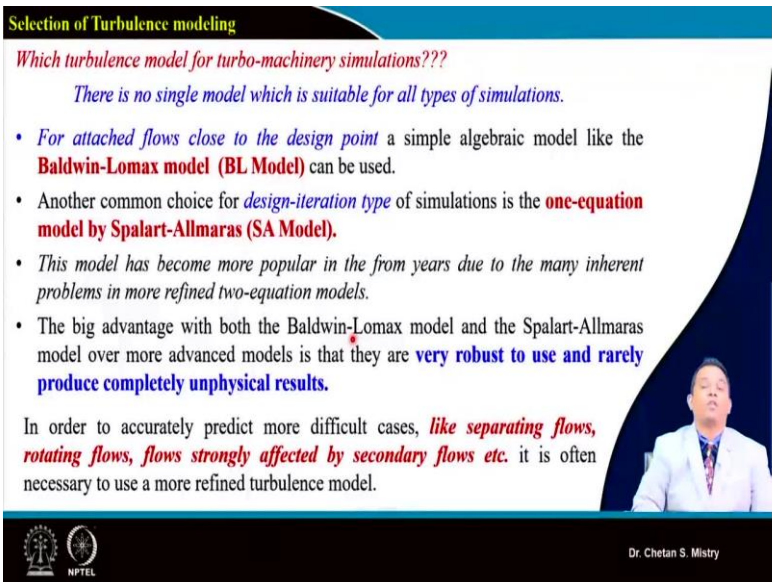
Here, in this case you can understand, this is what will be used for small or a moderate pitch, kind of configuration it is applicable for both single stage as well as for multi stage; you can see here, this is what is profile transformation. When we are talking about time transformation, we will be having exact exchange of data between this rotor and stator, okay.

Same way, when we are going with the Fourier transformation, that's what people these days they are using, in order to analyze say, aero-elasticity analysis, that's what is called blade flutter. So, when we are analyzing using this Fourier transform, we need to go with more number of flow domains, maybe 2 or 3. Here, in this case, single domain that's what is sufficient.

So, maybe if you are interested in more detail, you can go with the open literature available, which is discussing about this all aspects, okay. So, all these terms that's what is very important. These days' people, they are more interested in going with URANS, okay. Once they have done analysis, now their interest that's what is to do say unsteady analysis.

Now, in order to do unsteady analysis, you will be having one term, that's what is say time scale. Now, this timescale, there are different laws, that's what is applicable and this timescale what we are selecting, that's what will be based on what is the rotational speed of our rotor, okay.

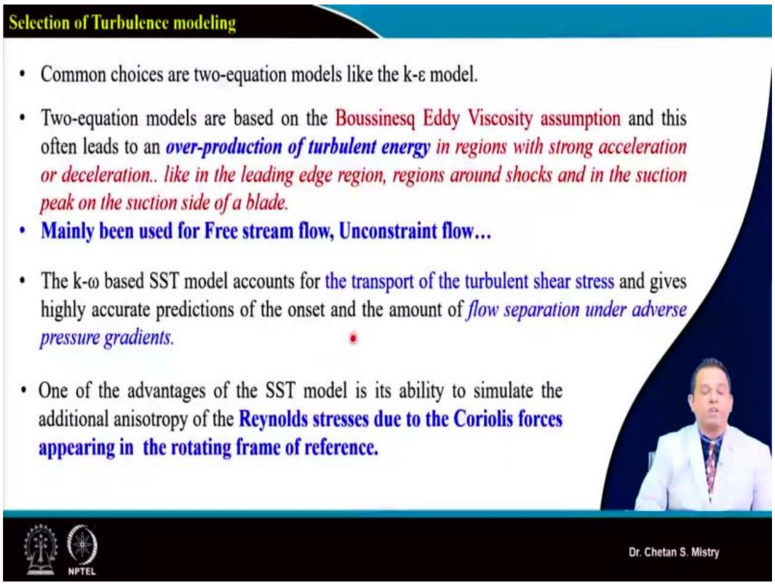
(Refer Slide Time: 28:34)



The next term, that's what is coming in terms of selection of say different kinds of turbulence models, okay. So, there is no single model, that's what will be used or we can say, that's what will be giving us the solution for which we are doing our analysis. There are different - different models, say first model it says, when we are having attached flow or say when we are having design kind of configuration, we can go with say, Baldwin Lomax model, BL model people they used to say, that's what is more than sufficient.

Sometimes, people, they are going with Spalart-Allmaras model or say SA model, it is called one equation model. So, this model, that's what is more popular, because it is more easy to use, and that's what is saving your computational time. Now, when we are looking for special kind of configuration, where we are looking for say specialized flow in terms of separating flow, maybe rotating flow or we are looking for secondary flow or we are looking for wake effect, under that configuration, this one equation model that will not help and for that we need to go with say two equation models.

(Refer Slide Time: 29:56)



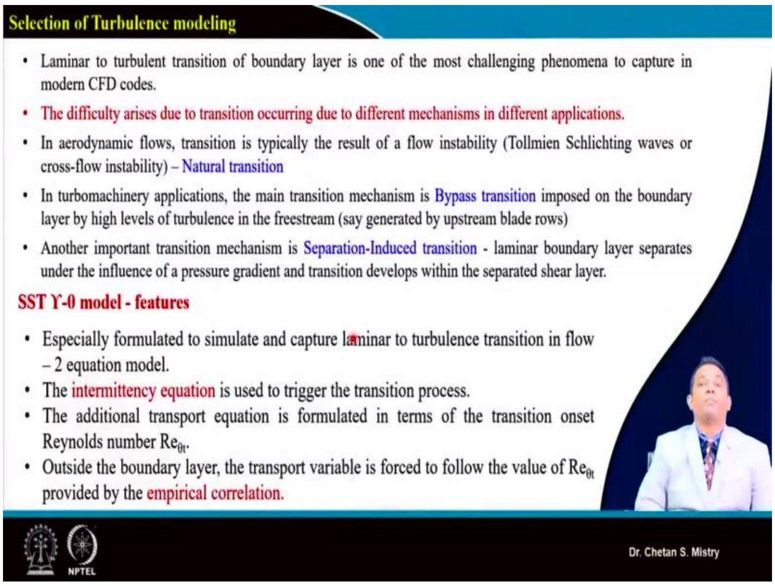
Now, what we mean by two equation model is we are having $k-\varepsilon$ model and $SST\ k-\omega$ model, we can say, that's what is say two-equation models. Now, this $k-\varepsilon$ model, that's what is based on the assumption of a Boussinesq Eddy Viscosity. So, when we are having this to be used for analysis of turbomachinery, it is giving access calculation of turbulence energy, because we are having flow it is accelerating somewhere it is decelerating somewhere, specially when we are talking about a leading edge.

Same way, we are having say shock formation, that's what is happening and my flow on the suction surface as we know, that's what is very challenging. So, under that configuration, this $k-\varepsilon$ model, that's what will not work fine. There are many aero applications; suppose if we consider, we are having freestream flow configuration or say unconstrained flow. So, when we are not having walls as a constraint, under that configuration, we can go with say $k-\varepsilon$ model.

So, it is not preferred; many times, people for initial calculation or initial analysis, they are using $k-\varepsilon$ model, but it is not preferred to go with say $k-\varepsilon$ model, because it has been defined specifically that's what is to be used for freestream kind of flow configuration. We have restricted kind of flow between two blades, we are having say constant flow configuration. So, under that configuration $SST\ k-\omega$ model, that's what is need to be used. It is capturing the flow separation under adverse pressure gradient correctly, that's what has been proposed by Menter.

And, the beauty of this SST model is it is analyzing say Reynold stress due to Coriolis component appearing in a rotating reference frame. So, here we can say, my flow when it is flowing through my rotor, we will be having say rotating flow as well as we are having sliding flow, we can say that's what is called say Coriolis effect and that need to be analyzed correctly. So, $SST\ k-\omega$ model, that's what has been preferred when we are analyzing say flow through axial flow compressor of flow through axial flow fan.

(Refer Slide Time: 32:21)



Now, we know what is happening on our suction surface of the blade. Initial part we know the flow it is laminar, after certain percentage chord my flow is going under transition condition and later on that's what will be going under turbulent condition. Now, what all model we have discussed, say Lomax model or maybe S-A model or $k-\varepsilon$ model, $k-\omega$ model, they are unable to capture the flow transition and that flow transition it is of different kinds, we can say it is a natural transition, bypass transition or maybe separate separation induced transition.

Under that configuration, it is very difficult to understand what is happening on the blade suction surface. Because how my transition and from where that transition is occurring, that's what is of our great interest, because that's what will be deciding how my flow will be behaving towards the trailing edge.

Now, in order to analyze such kind of flow, there is a new model, people they have proposed, it is called $SST\ \gamma-\theta$ model. Now, this is what is capturing the transition flow, that's what is required, the intermittency equation that's what has been solved or that is called when we are talking for analysis purpose.

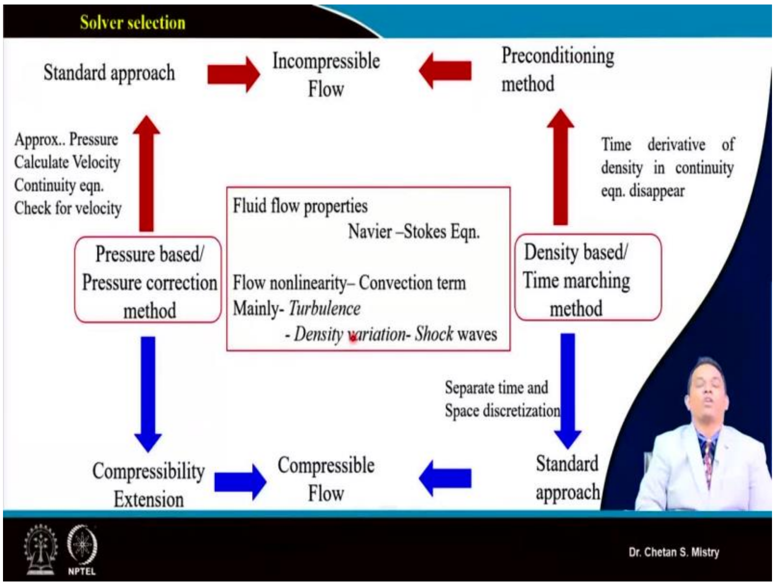
So, basically the transition onset Reynolds number it is calculating after particular locations, it has been forced to use this that will give the solution with say transition flow that is happening on the blade surface.

So, over the time with maturity and understanding, people, they are doing their analysis. Now, the question always is what to be used, but the thing is what we want to analyze, that's what will be deciding what parameter need to be selected with. So, now, we are having different turbulence models.

Now, let me tell you one more parameter, that's what is called turbulence intensity. Now, say when we are doing our analysis, suppose say cascade tunnel when we are analyzing, it may be having low turbulence effect. Now, this turbulence effect, that's what will be bringing the change of this transition on the surface. So, mainly when we are doing our analysis, say initial stage of analysis, this turbulence intensity we are assuming 5%.

In actual engine also, this turbulence intensity, that's what will be ranging in the terms of say maybe 5 to 10% of turbulence intensity; what kind of flow it is flowing inside my engine, that's what is basically giving us idea in sense of turbulence intensity. And, this turbulence intensity, that's what is varying with different kinds of flow configuration, different kinds of flow structure, that's what is happening between the stages. So, we need to be very careful which stage we are analyzing.

(Refer Slide Time: 35:33)



Now, the question that's what will be coming next is in terms of selection of solver. So, basically, what we are doing? We are solving say Navier-Stokes equation, in order to calculate my flow properties. Now, we are having flow, non-linearity that's what we say, as a convection term, that's what is because of maybe turbulence, or maybe density variation because of presence of shocks.

So, now the question is, how do we or what do we select in terms of solver? So, there are two different kinds of solvers, people they have defined with; one, that's what is pressure based solver, or we can say pressure correction method; second, that's what is say density based solver, or we can say it is time marching method.

Now, this density based solver, that's what is a standard approach when we are analyzing our compressible flow. Now, when we are going with say incompressible flow, the standard approach it is to go with, say, pressure kind of solver.

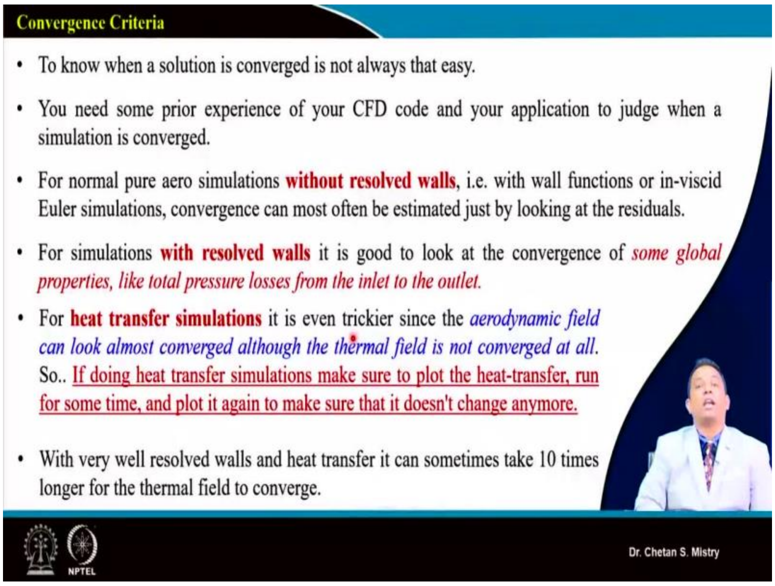
Now, this pressure and density based solver, it is always a question mark, with say to a code developer as well as end users. Someone will say, no, I want to use my density based solver in order to solve incompressible flow rather than compressible flow, then we need to go with say, time derivative term for this density what we are using in continuity equation, that's what will be getting say nullified or it will be disappearing and we will be using preconditioning method.

Now, suppose if you are looking for standard approach, where we are having separate time and space discretization, that's what approach we are adopting for. When we are talking about say pressure based solver, we initially put say approximation of pressure, then based on that it will be calculating the velocity based on the continuity that need to be satisfied, it will check with say different velocity configuration.

Again, the question will come; now, I want to use pressure bases solver for say compressible flow solution that is also possible. Under that condition, we need to go with say compressibility extension. Now, this is what is not the course of CFD.

So, again and again, I am saying you go with say basic course of CFD or maybe advanced level course of CFD, in which people they are discussing about the development of solver, different numerical techniques, how do we solve our Navier-Stokes equation, that's what is very important. But, for initial basic understanding, for kind of analysis what we are looking for, this knowledge, that's what is sufficient at this moment.

(Refer Slide Time: 38:28)



Now, say one more term, that's what is coming is say convergence criteria. Now, what analysis we are doing, we are more interested in terms of whether my solution is correct or not. Now, that's what will be decided based on say convergence criteria. So, based on the judgment, people they used to analyze these results.

So, residuals, they will be calculating for say continuity equation, momentum equation, even for energy equation. Now, Best Practice Guide, it says maybe when we are analyzing say resolved walls kind of configuration, suppose say when we are analyzing flow to axial flow compressor or say fan, we need to put our observed parameter, suppose one of the parameters you can say total pressure at the outlet.

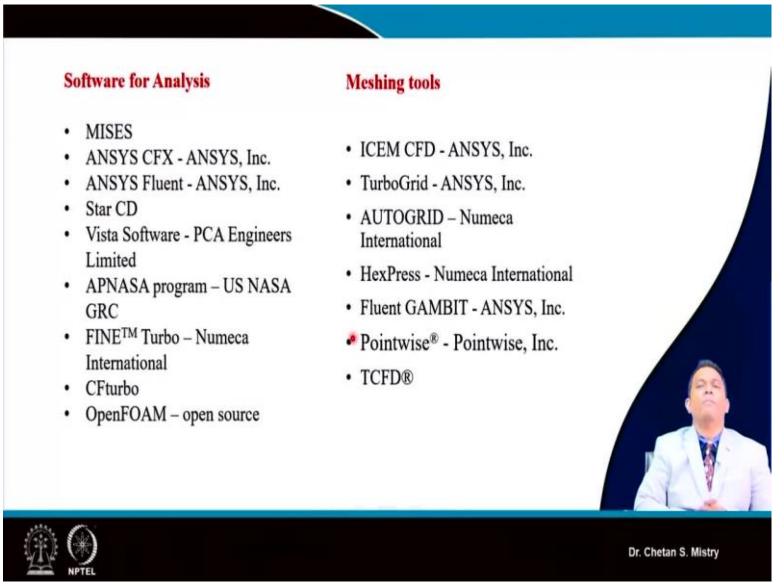
So, when you are putting that as a observed parameter, the fluctuation, that's what will get steady. So, once it will be reaching to certain limit, it will give the constant value, we can say, my solution it is converged. So, that is one of the way. When we are analyzing the heat transfer, say gas turbine cooling problem, that's what is very challenging. What will happen, there we are analyzing our continuity equation say that's what is talking about say aerodynamic aspect that will get converged.

But when we are analyzing for say heat transfer, it may be possible that my solution will not converge, under that configuration, we need to stop this simulation, again you put your observed parameter, when you are getting that number to be constant you can go with say the solution it is converge.

Now, such kind of analysis that's what is required whole lot of time for you, okay. So, there are best practice guides available, for that people, they are using say RMS value, root mean square value, and as a residual in order to observe how my convergence that's what is happening; we used to say $10^{-4}, 10^{-5}, 10^{-6}$, that's what is the residual value. We can say my further change of debt parameter that's what will not be happening.

Now, number of iterations, be careful! There are two terminologies we are using; one, it is residual part, that's what is say convergence criteria, we say 10^{-5}, it is my convergence to be expected with, second, that's what is number of iterations? Suppose say, I have defined 500 iterations. So, when my solver that's what is running, it may not reach to $10^{-4}\ or\ 10^{-5}$ residual, but my number of iterations what I have defined, it has reached. And, that's what will be stopped your solver to move forward. Be careful, you always keep an eye with say some observed parameter for best practice or best solution.

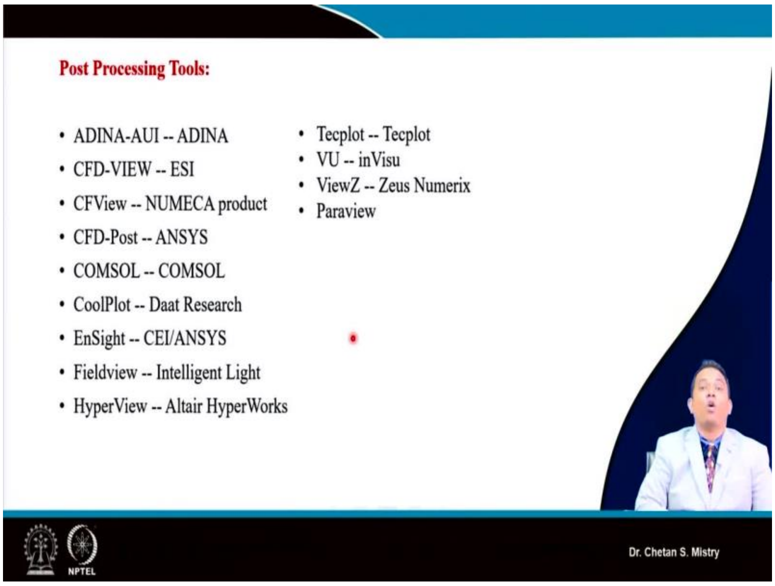
(Refer Slide Time: 41:30)



Now, there are many software which are available in open domain. They are say commercial or maybe freeware. Say, suppose if we want to analyze our flow domain, we are having say MISES, ANSYS CFX, ANSYS Fluent, Star CD, Vista software, APNASA, FINE Turbo, CF turbo, Open FOAM, that's what is say open source. So, there are many solvers which are available. We can say solver is nothing but that's what is the software, okay. These are the commercially available or freeware available solvers, okay, for solving our flow field for axial flow compressor as well as for axial flow fan.

At the same time, in order to do discretization, we say for meshing, we are having ICEM CFD, TurboGrid, AUTOGRID, HexPress, maybe fluent GAMBIT, Pointwise, TCFD; these are different software which are available. You can use one of the tool for discretization of the domain.

(Refer Slide Time: 42:36)



Now, very important, that's what we need to understand is what we are doing in terms of simulation, what results we are getting, that's what is called post processing. That post processing, we are having different kinds of software available say ADINA, CFD VIEW, CFview, CFD post, COMSOL, maybe, CoolPlot, EnSight, Fieldview, HyperView, Tecplot, VU, ViewZ, may be Paraview, all these software which are commercially available or freeware available for analyzing or doing the post processing.

Now, doing post processing that is also a skill. So, in concluding part for today's session, we can say we started talking about different boundary conditions. After those boundary conditions, we were discussing about different interfaces that need to be selected between rotor and stator for analyzing our flow through the stage.

Then we were discussing about the selection of different turbulence models. Then, we were started discussing about say different kind of convergence criteria, that's what we need to keep on eye. And finally, we have discussed about different kinds of software, they are being used for solving maybe for meshing or for doing post processing.

So, here we are stopping with. Now in next lecture, we will be discussing about say different case studies based on what all we have discussed up till now. Thank you, thank you very much for your kind attention!