Turbomachinery Aerodyanmics
Prof. Bhaskar Roy
Prof. A M Pradeep
Department of Aerospace Engineering
Indian Institute of Technology, Bombay

Lecture No. # 39

CFD for Turbomachinery:

Flow Track and inter-spool Duct Design using CFD

Welcome to lecture number 39 of this lecture series on turbomachinery Aerodynamics. We are approaching the fag end of this lecture series, which has been journey into discovering various aspects of turbomachines, which are relevant to an aircraft engine. We had discussions, quite detailed discussions on different types of compressors, basically devoted towards the axial type of compressors and some amount of discussion on the centrifugal compressors. We also discussed in detail about axial turbines and the radial turbines.

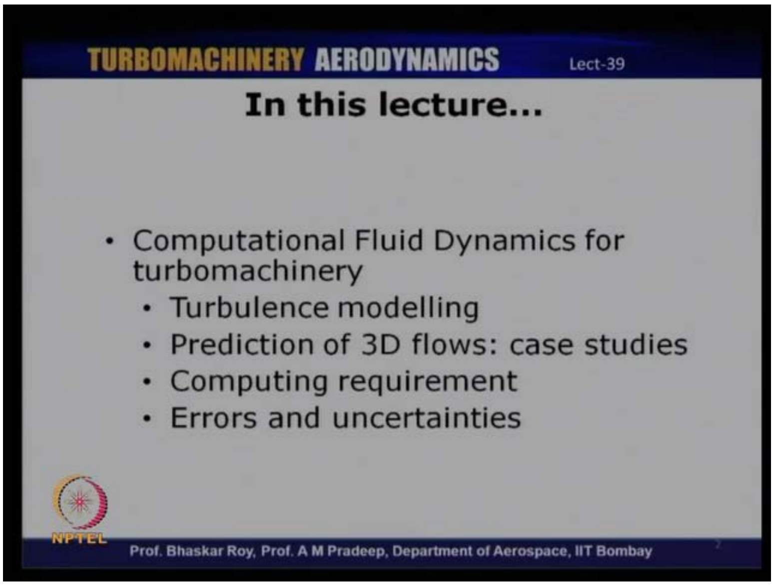
So, the last lecture, today's lecture, and the next lecture, are exclusively devoted to discussion on certain aspects of Computational Fluid Dynamics for turbomachines. Those of you who are aware of CFD, you probably are aware that CFD is a very powerful design tool, which has emerged as a third option of analysis of engineering systems. We had some end up discussion on this in the last class.

So, the other two options that where existent for a long time or experimental methods and analytical or theoretical methods. CFD is a third alternate approach, which has which is being currently used worldwide by a variety of engineers looking at a variety of different applications. Turbo machinery flows happen to be very complex and therefore use of CFD for turbo machines though it is very rampant in the design stage, there are lot of issues which are still which still need to be resolved, before CFD can actually be considered to be a very standard design methodology, though it is still being used as a very widely for design practices.

There are lot of issues associated with CFD and it's capability in predicting certain aspects of flow, especially towards which which are to do with off design operating conditions; certain characteristics like the instabilities, install and search are not very well captured by CFD. So, we will try to look at some of these issues in today's class, where I will also be taking up a few case studies, where a certain few researches have actually been able to use CFD to map very complex flow fields and have very convincing validations of this CFD results with experimental data. So, we will also look at a few sample results from papers which have been published in the last 3,4 years, which have been proven which have sort of proven that CFD can actually be used in very in in turbo machine flows with which are as we know very complex and three-dimensional, highly turbulent and so on. CFD has been proven that it can be used with a of course, a lot of caution that needs to be taken while using CFD.

So, today's class, we will discuss about a few aspect which are still, which which still need a lot of research and understanding that is to do with what are known as turbulence models, and we will talk about the different types of turbulence models that are existent, and what are the advantages and disadvantages of each of these different types of turbulence models. We will spend quite some time discussing about turbulence models, then we will look at how CFD has been able to predict very complex structures in in a turbo machine flow, we will be talking about the use of CFD in predicting complex 3D flows. So, we will also look at some case studies in today's class, towards the end we will also have some understanding about what are the pitfalls involved in CFD study, just like in experimental analysis, an experimentalist always gives a certain error margin and uncertainty band in all the results that it produces. Similarly, is there a certain error band that a CFD user can actually associate with any of his results that something that we will discuss in as one of the topics in today's class.

(Refer Slide Time: 04:46)



So, we will basically be talking about these topics, we will start with Turbulence modeling. We will talk about Prediction of 3D flows; we look at lot of case studies. We will have some discussion on computing requirements and the ever growing computing requirements year after year and of course, we will also be talking about Errors and uncertainties in one of these slides. So, when we let us take up the Turbulence modeling first. Now Turbulence modeling and what are its implications, why do you need turbulence modeling is what we shall be talking about. So, before we take up that you probably aware of the fact that certain averaging techniques are used in trying to simplify the navier-stoke equation.

So, Reynolds averaging is one of the most common ways or common methods by which an averaging can be done; on the navier-stoke equation to kind of simplified and make it more amenable for analysis and this averaging can be over different phases, it could be either averaging in space or it could be in time or it could be an un symbol average. So, Reynolds averaging could be in either of these components, it could be in space time or a Reynolds able average. The basic idea being in averaging is that you would like to separate out the average flow property from the mean or fluctuating property. So, in a turbulent flow as we know, you can, you also have a certain fluctuating parameter associated with let us say velocity or time. So, you would also like to consider those parameters separately.

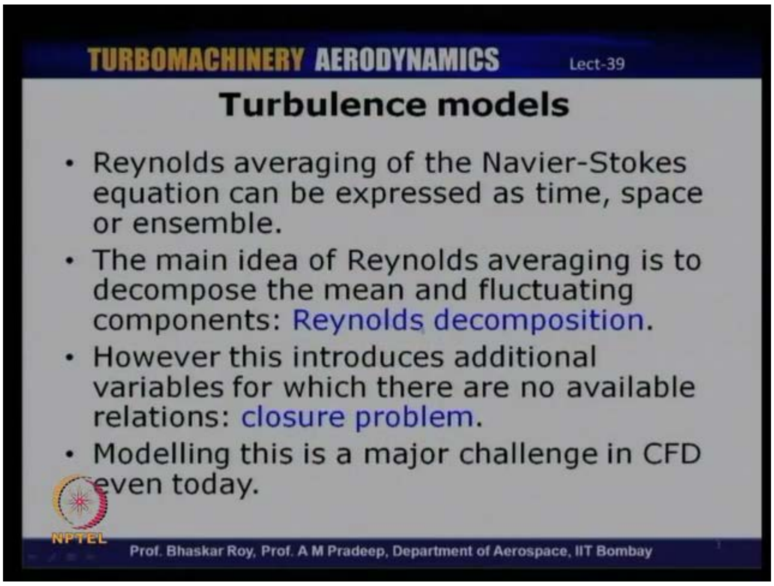
So, Reynolds averaging provides us a method of decomposing or decoupling well not really decoupling, but at least decomposing the flow into its mean and fluctuating components. This is also refers to as the Reynolds decomposition, but by doing this, what essentially happens is that is introduces a new set of variables, which are basically the Reynolds stresses which cannot really be solved. So, you will basically look at the different governing equations and you see the number of unknowns, you will have more unknowns than the number of governing equations themselves.

So, how do you solve this set of simultaneous equations when you have more unknowns than the number of equations? So, this produces what is known as a closure problem that is you cannot really close this set of equations, because you have more unknowns than the number of equations themselves. So, in modern day CFD, it is a practice that one would model the terms, which are unknown; and these methods where various methods by which one can model these unknown parameters and that is essentially known as turbulence modeling.

So, turbulence modeling basically involves modeling some of these unknown parameters, which are a basically a consequence of the Reynolds decomposition, leading to a closure problem and by means of certain empirical means can we model these different parameters. So, that is done by using turbulence modeling, and it is been found that this parameter that we are trying to model is not a function not necessarily a function of of the fluid, but it is essentially a function of the flow itself for the flow field itself, which means that depending upon the nature of the flow these parameters can keep changing and that is where the major challenge lies in how do you model something the behavior of which is very uncertain, because you do not know, how the flow is going to behave, because this parameter which you are trying to model is not a function of the fluid, but it is a function of the flow.

So, what you will find if you have used if you have had a change to use a commercial CFD package, you will find that there are various options for turbulence modeling. So, all these options have been provided because these turbulence models are not necessarily universal that is you cannot use one turbulence model for all kinds of different applications. There are lot of limitations which each of these turbulence models have which is why there are certain applications which are suited for certain types of turbulence models and so on. So, one need to be very careful in deciding what kind of turbulence model that you would need to use for a particular application.

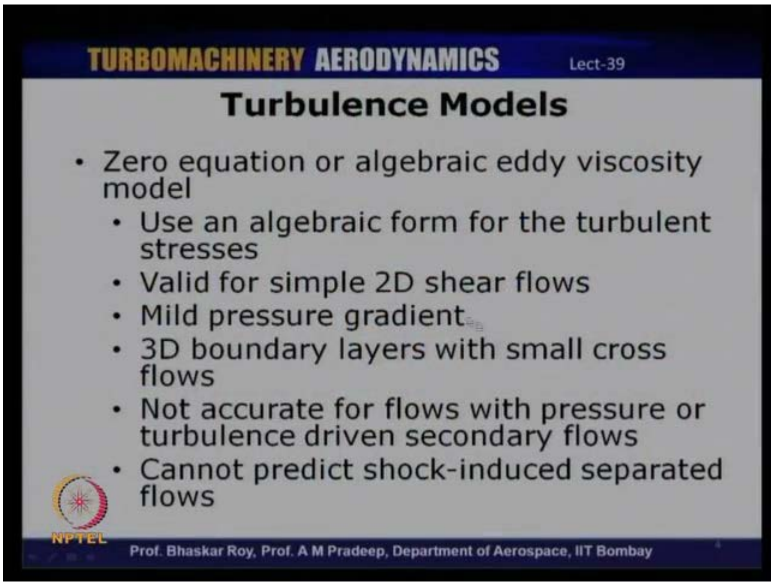
(Refer Slide Time: 09:31)



So, Reynolds averaging, as I mentioned is primarily intended to decompose the mean fluctuating components which leads to a closure problem, because there are it leads to certain additional variables for which you do not have any amenable relations. So, how do you model this and this becomes a major challenge in CFD even today and lot of researchers all over the world are still struggling to figure out, what is the best way of trying to model is there a turbulence model which can be used as a universal turbulence model, which is independent of the type of flow that we are dealing with. Unfortunately at this movement, we do not really have turbulence model, which can be said to be a universal turbulence model which can be applied for any kind of flow. So, let us start with the simplest type of turbulence model; simplest type of turbulence model involves approximating these unknown parameters, the Reynolds stresses in terms of an algebraic stress or algebraic eddy viscosity, which means that you are simply considering that as a term which is very similar to the normal fluid viscosity, and that is why such turbulence models are called zero equation or algebraic eddy viscosity model.

So, these algebraic eddy viscosity models essentially, uses an algebraic form of the turbulence stress, and therefore it is trying to approximate this stress term, additional stress term using certain algebraic expression. It has been found that these are very simple turbulence model, and they do not consume too much of computing power, but they are essentially valid for very simple two-dimensional flows. In the presence of more complexities and when it is used for three-dimensional flows, then the flow this turbulence model does not really predict the flow very well.

(Refer Slide Time: 11:34)

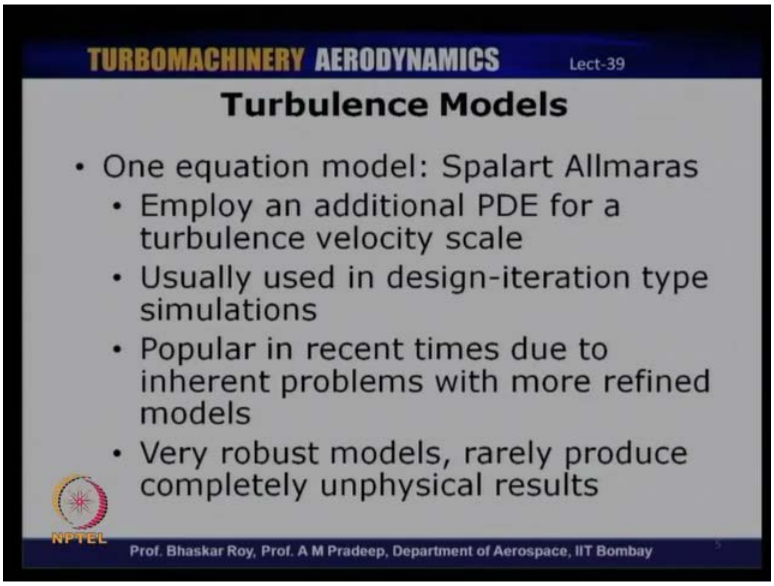


So, a zero equation or algebraic eddy viscosity model, it has been used for simple 2D shear flows with very mild pressure gradient, may be when 3D boundary layers with a very small cross flow; whereas, if you look at flows which have, which are highly turbulence driven, and which are secondary flows and which may have shock-induced separated flows and so on; the zero equation or algebraic viscosity eddy viscosity model simply fails and they they have not been really used in with such applications with lot of success. So, that is puts a lot of limitation on this type of a model, which is very simple to begin with its very simple model to use and it does not really require lot of computing power.

So, if you are dealing with very simplistic two-dimensional flows like say for example, a cascade flow perhaps the zero equation model might be able to give you some reasonably good results, whereas if you look at very complex 3D flows which involves secondary flows and tip leakage flows and so on. Zero equation models simply cannot be used. So, then we have what are known as one equation model, one of the most popular one equation model which has been developed is known as the Spallat Allmaras equation or SA model and what is this model basically involves the use of an additional partial differential equation which is used for the turbulent eddy viscosity or eddy stress.

So, the turbulent stress or velocity scale is represented by an additional PDE and therefore, that is why it is called single equation turbulence model, and this Spalart Allmaras model again is very simple, and it has been found to be very robust for wide range of applications. So, for a variety of applications, this Spalart Allmaras model has been used and demonstrated to be quite successful.

(Refer Slide Time: 13:37)

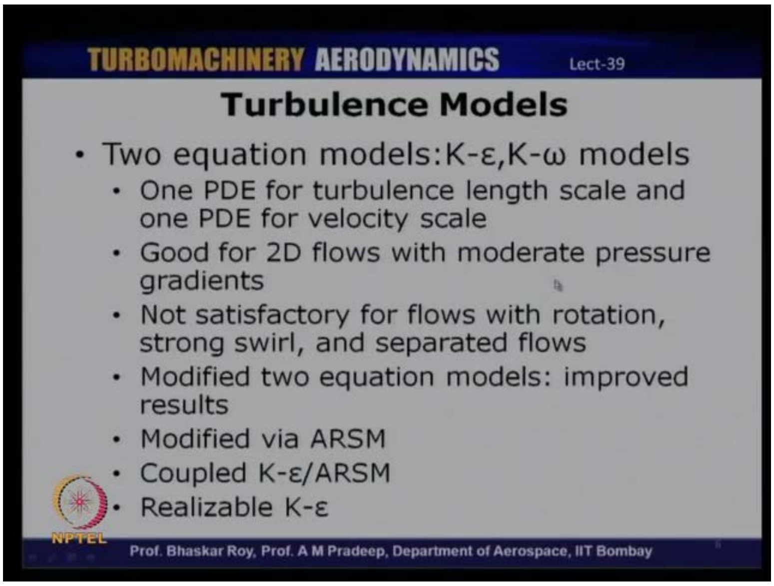


And so, in this case it is primarily inducted to be used for the design- iteration kind of simulation not for very detailed studies. It is kind of becoming very popular in the rather recent times because of problems associated with the other type of more complex models. There are many more complex models that I shall be discussing about those models have lot more complexities and issues and therefore, that makes this model little more attractive and this can be used in wide range of applications.

It has to be it is has been found to be quite robust in the sense that it has been able to predict results in variety of applications in some of them it is not the results may not quite correct, but it is not truly unphysical as well. There are certain turbulence models which in some flow situations actually produce unphysical results which makes it very difficult to decide what kind of a turbulence model it should be using Spalart Allmaras model in that sense is a safe model that can be used in a variety of application.

Now, if you look at a slightly more complex model, we have the two equation models, where there are two partial differential equations; one is used for model for for the turbulence length scale and other for the velocity scale. So, there are two PDEs, which are used, additional PDEs over and above the governing equation, which are use to model various, one is for turbulent length scale and the other is for the velocity scale. So, there are two common types of these two equation models which are used, one is known as the K-epsilon turbulence model and other is called the K-omega turbulence model, these are again used with and without certain additional features.

(Refer Slide Time: 15:29)



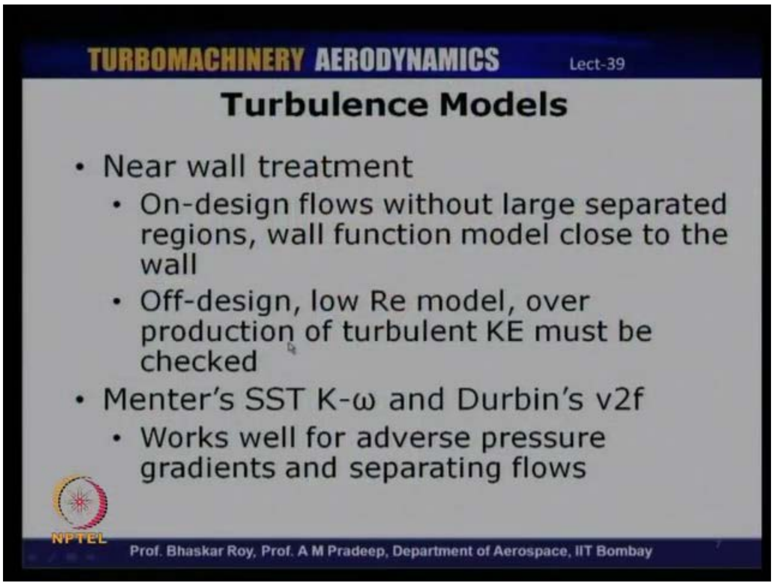
So, if you look at the two equation turbulence models, these are primarily good for 2D flows with a moderate pressure gradient. The problem with these flows are again the fact that these turbulence model is the fact that these models again do not produce satisfactory results, when there are when there are rotational effects or swirl or separated flows; and there are of course, modified versions of these turbulence models, which are the Reynolds Stress Models or Coupled K-epsilon and Reynolds Stress Models and realizable K-epsilon models and so on, which are basically modifications of the basic K-epsilon or K-omega models, which can be sort of used in certain set of applications. But again as a set, one of the main limitations of these two equation models is the fact that they have not been really proven to be very effective in flows, which involve large amounts of separation or with rotation or swirl.

So in highly three-dimensional flows, even the two equation turbulence models like the K-omega or K-epsilon or not really satisfactory and especially, if you look at a compressor, which is operating very close to stall. The flow becomes highly three-dimensional and unsteadiness begins to creep in in such flow situations the K-epsilon, K-omega models need not necessarily predict the flows very well. So in in such situations, the basic problem associated with these models is, how do you kind of understand the dissipation of vertices, which takes place very close to the boundary layer. So, in order to do that one needs to resolve the boundary layer very carefully and therefore, near wall treatment of the flow becomes very significant and so, the results become quite sensitive to the type of grids which are used very close to the surface and that is where the near wall treatment or wall functions become very important.

So, if you look at the basic two equation models, and try to control understand their functioning with reference to different flow fields, they have certain inherent limitations as I mentioned. So, if you look at let us say K-epsilon model, the use of K-epsilon model, for example, diffusing separate at flow is going to give you highly the the results which are going to be highly different from what it should have been, it does not predict the flow physics very well. Basically what happens is that the rate of change of production of turbulence kinetic energy is quite different, and it is a strong function of the flow field itself and that is something that is being modeled by these two equation models.

So, unless those flow physics are properly modeled in these by this turbulence model, it can obviously, lead to lot of unphysical results. So, it in this context that there two other modifications to these or two new types of turbulence models, which have been proposed in the last few years which have been found to be quite satisfactory and being able to predict flows in pressure gradients as well.

(Refer Slide Time: 18:56)



So under off-design condition, low Reynolds number model what basically happens is that there is an over production of turbulent kinetic energy, which is predicted by some of these model. The two other models which are gaining popularity are the shear stress transport K-omega, which is which was proposed by Menter. So, it also called the SST K- omega model and the other turbulence model is the Durbin's v2f model volume of flow model and these have been shown to be quite effective and it it is seems to predict flows with adverse pressure gradient as well. So this, these two models seem to have or seem to show lot of promise over some of the conventional turbulence models which have been used over several years.

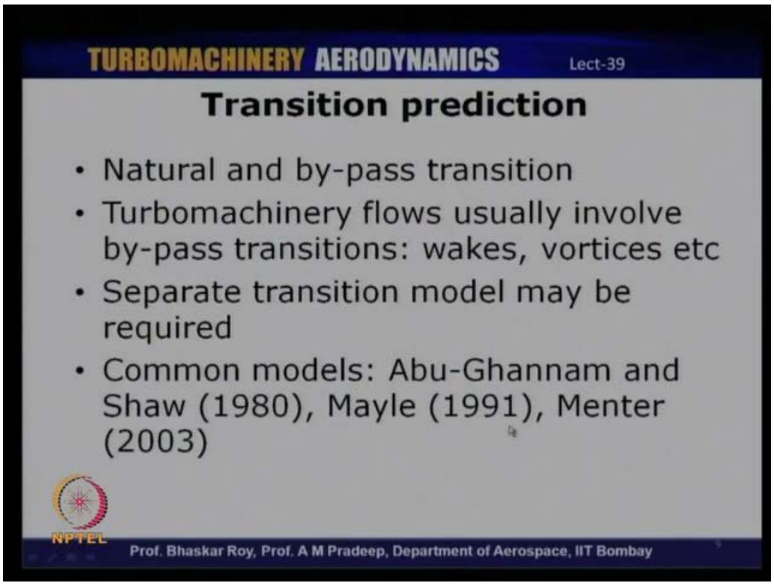
So, in SST K-omega and Durbin's v2f model and some of the modern day commercial codes have options of these new turbulence models as well, and therefore that gives the designer CFD user, a lot of flexibility in trying to choose what is the best model that is suited for his application, for example, if a CFD user is trying to look at very simple flow in a channel or a duct which does not have a great deal of pressure gradient, and let say, it is two-dimensional or axis symmetric, he does not really have to go for a high end model like the SST K-omega or v2f, a simple Spalart Allmaras one equation model or even the algebraic stress model that is the zero equation model also would be able to predict the flows reasonably well.

So, it depends upon the kind of application that one is looking for that based on which, one can decide what is the kind of turbulence model that one needs to use. Let us now, take a look at a few other turbulence models, which are multiple equations models, let us say for example, the Reynolds Stress Models usually they use up to seven partial differential equations to model each of the components of the turbulence stresses. So, there are several different equations which are used, which means that the more number of equations one uses, the computing power required for such using such a turbulence also is going to be higher and it has been found to be reasonably well behaved in a different variety of flows, the main issue being the amount of computing power that is required for designer to use this kind of turbulence model.

So, Reynolds Stress Models it basically uses up to seven different partial differential equations, for all the different components of the turbulence stresses; and it is found to be reasonably better in flow situations where the other models, especially the two equations model were unsatisfactory; and it is being found to predict the turbulence flows in more realistic fashion than some of the other turbulence models which have been used. So, Reynolds Stress Model is high end, I would say high end turbulence model, but not necessarily the best, because of the fact, that it requires a lot of time to use the Reynolds Stress Model where in up to seven additional PDEs are introduced trying to solve each of the different components of the turbulence stresses. So, if you are looking at Reynolds Averaged Navier-Stocks equations, the ramp simulations as I had discussed in the last class, turbulence model is going to be a major issue. So, depending upon the kind of application that one is interested in the type of turbulence model needs to be chosen accordingly.

Now, there are other issues involved here as well. Now, one let us say we have been able to use some turbulence model with reasonable accuracy, the next major bottle neck is the fact that if the flow involves a transition, it is either natural transition that is laminar to turbulent transition or a by-pass transition, whatever be the case, which is quite rampant in turbulent flow, in a turbo machinery flow, which has wakes and vertices and bluff bodies and so on. So, in the presence of all these, the by… You may have encounter either natural transition or most often you might also involved by-pass transition. So, if it is by-pass transition, then one needs to introduce or use different type of model to model this transition itself, how do you take care of transition, if let us say, it is a by-pass transition; it is not a naturally transitioning flow from laminar to turbulent, it is a by-pass transition. So for that again, there are turbulence or models which are used for transitioning or transitioning flows.

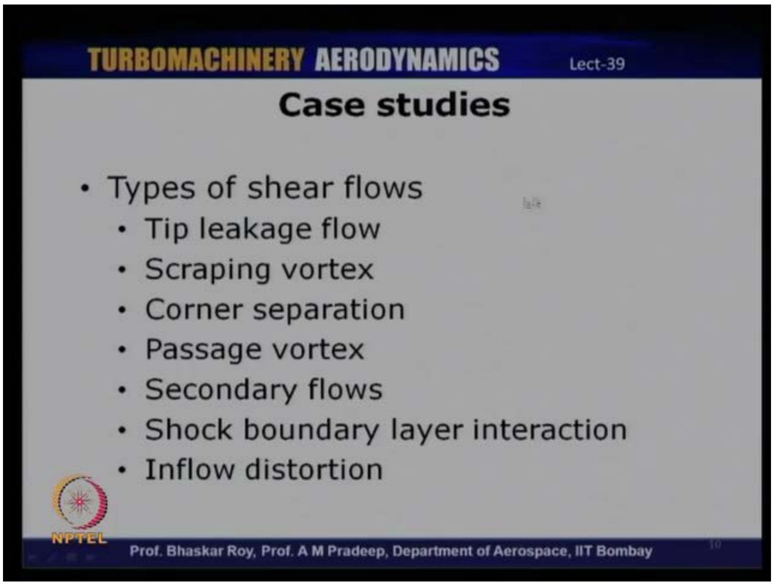
(Refer Slide Time: 24:10)



Some of these common transition models are by Abu-Ghannam and Shaw which was proposed in 1980s, then we had a transition model by Mayle in 1991 and the most recent one being attributed to Menter in 2003. So, these are the different types of transition models which one can use, if the flow that we are looking at indeed has a elements of transition present, one needs to model transition as well there are very well let out models for laminar flows and turbulence flows and while turbulence flows are of course, one has to model turbulence transition model also becomes quite significant, in these different types of flows where one might encounter certain elements of transition.

So, having understood one major bottle neck or stabling bock in in CFD simulations which is the Reynolds the turbulence model, let us now take a look at some case studies where in very complex 3D flows which are associated with turbo machinery flows have been simulated and validated fairly well with experimental data. We will take a look at at least three or four different these case studies where people have published their CFD results which they were able to predict very well and compare their results with experimental data.

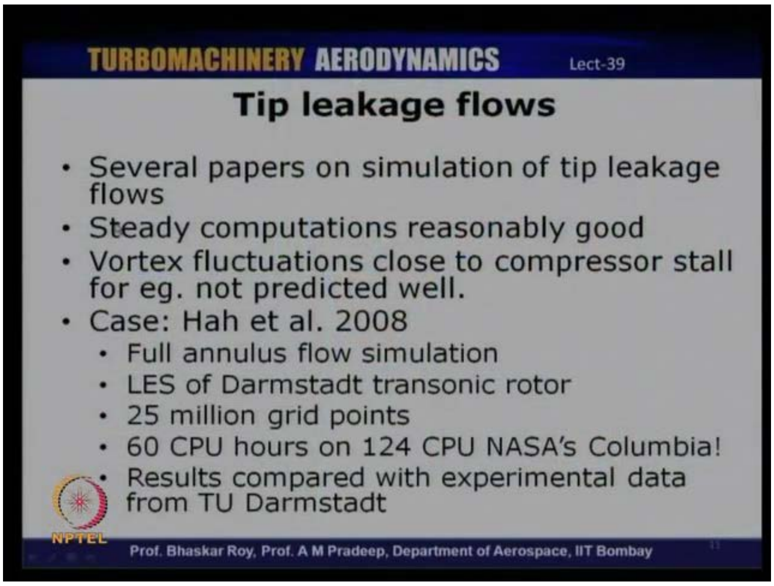
(Refer Slide Time: 25:37)



So, in a typical turbo machinery flow as we are aware, these are the different types of shear flows which are possible; you may have tip leakage flow this is on account of the tip gap between the blade and the casing, and the pressure difference between pressure surface and suction surface. One may have scraping vortex which is because of the motion of the blades and removal of the boundary layer from the casing, leading to a vortex which is a scraping vortex.

One may have corner separation or passage vortex, which is basically the vortex which gets trapped in the passage, and you may have in addition to that secondary flows and or you may have shock boundary layer interaction, inflow distortion and so on. So you can see that there are these different complexities that get involved, when you talk about a turbo machinery three-dimensional 3D turbo machinery flow. So, to be able to predict these different flows very well, one has to make sure that this CFD scheme that one is using is indeed correct and lot of literature you would find which have been published last couple of decades have been devoted towards trying to improve CFD models to be able to predict these different complex flows in an accurate manner and so, you will find in literature there are numerous paper on each of these different types of simulations.

(Refer Slide Time: 27:26)

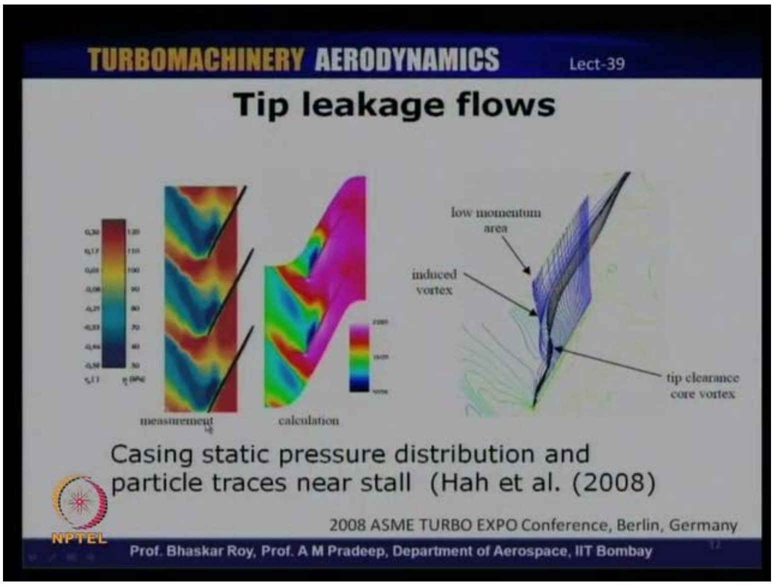


Let us take up the tip leakage flow first. Tip leakage flow is one of the most highly thoroughly explored areas in the turbomachinery flow and research area and you will find lot of literature published in various journals and conferences and what is been observed is that in tip leakage flow, the steady computations are reasonably you do not really need to go for an steady simulations, but when it comes operation near compressor stall then that is really not well predicted.

Let us take up our first case steady first, this is by a group it it at Nasa Langley by researcher name Shunil Hah and his group which was published in 2008, who did a full annulus flow simulation, and then basically they carried out a large eddy simulation LES of Darmstadt transonic rotor, Darmstadt is university in Germany where they have a transonic compressor testing; so, this group at Nasa Langley carried out a LES of the Darmstadt transonic rotor, and let us look at the numbers which are involved here.

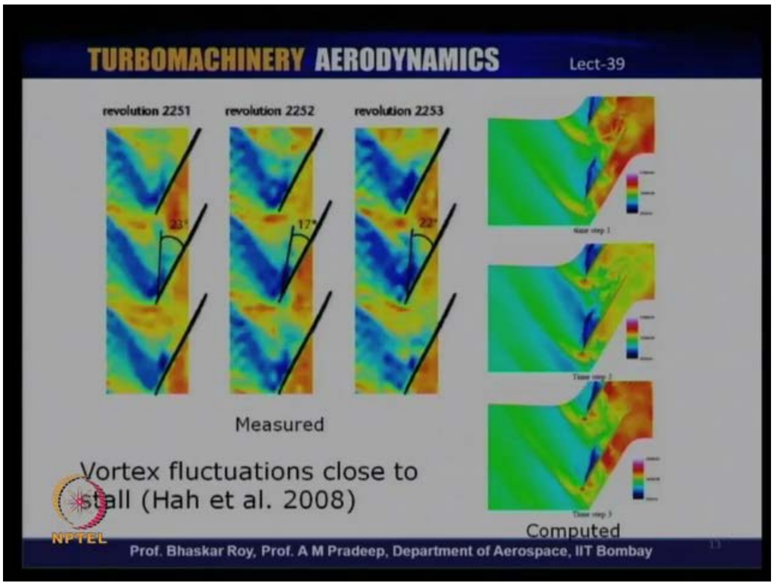
This kind of a simulation required 25 million grid points and the simulation required 60 CPU hours on 124 CPU NASA's Columbia, which is server, which they have a high end high performance server, 124 CPU's of these servers 60 hours on that for a 25 million grid point simulation and this CFD data was compared with experimental data from the Darmstadt casting. We will now that a look at how this CFD results compare with the experimental data which was observed which was obtained from the same casting.

(Refer Slide Time: 29:13)



So, if you look at the CFD data and compare that with what was measured, so here, we have the measurement data from the casting and these is the CFD or calculation data and what is plotted over here is the casing static pressure distribution and particle traces near stall. You can see an excellent match between the CFD and measurements, you can see that the low pressure reason has been very well captured in both the CFD as well as in the experiments and it matches quite well. So, from the CFD data they could figure out the clearance, the core of the tip clearance vortexes as you can this is the tip clearance vortex a low momentum area and an induce vortex which has been quite detail captured in quite detail by this CFD steady and by this result was of course, published in the TURBO EXPO conference, which was held in Germany in 2008.

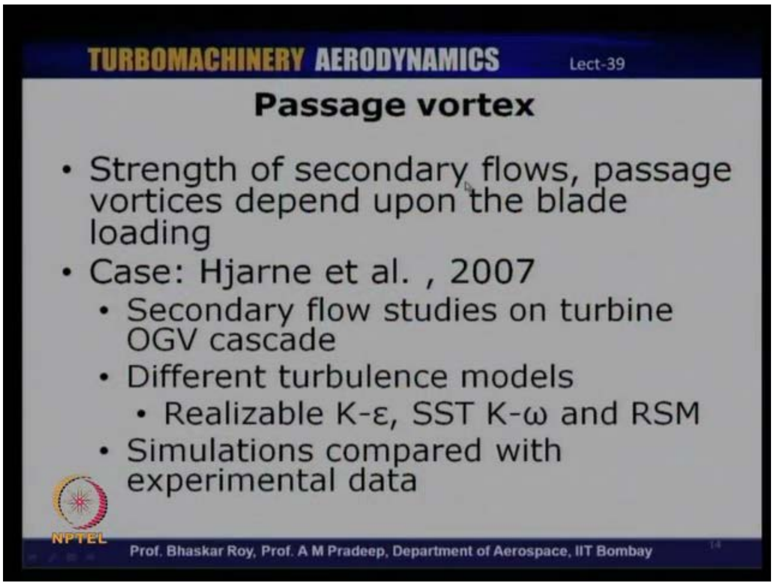
(Refer Slide Time: 30:13)



So, this is taking it further, let us also look at vortex fluctuations which was because it was large eddy simulation, there is a transient element already present it is time matching solution and so, you can see that as the rotational speed increases what is been measured is the fluctuation in the vortex itself. You can see the angle shown here as 23 comes down to 17 and then it again goes to 22. The same thing exact same thing as been captured by CFD solution as well were you could captured the vortex fluctuation in a reasonably accurate manner.

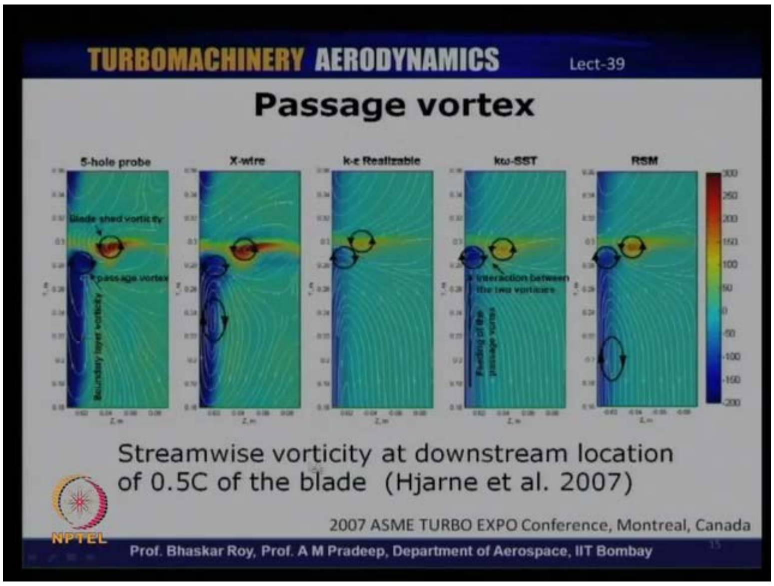
So, what is seen here is that even though the flow is highly complex, it is transonic which means there is a shock boundary layer interaction taking place use of LES which is larger dissimulation which is much more consider to be much more higher accuracy solution as compare to the ramp simulation has been able to predict the performance even with even with reference to what is it if or vortex fluctuations and that is of course, the amount of time and effort involved in this was huge, but even then it has been able to predict the performance reasonably accurately. So, that shows the kind of power or strength which CFD has in terms of trying the way we are progressing towards in in terms of our capabilities in terms of simulating very complicated flow fields. So, what we have looked at was one case that was the tip leakage vortex. We will now take a look at another flow field scenario which is essentially known as the passage vortex which is again highly complex, but it is not as unsteady as one would expect tip leakage vortex to be.

(Refer Slide Time: 32:12)



Now here we would look at passage vortex, the strength of secondary flows and passage vortex primarily depend upon the blade loading and this is basically study which was again published in 2007 by a group called Hjarne and his group, where they looked at secondary flows on turbine outlet guide vane cascade. So, they have used different types of turbulence models to sort of compare the performance of each of them, they have used K-epsilon model with some wall corrections the realizable K-epsilon model, the SST K-omega, the Menter's SST K-omega and the Reynolds Stress Model and the simulations have been compare with the experimental data.

(Refer Slide Time: 33:01)



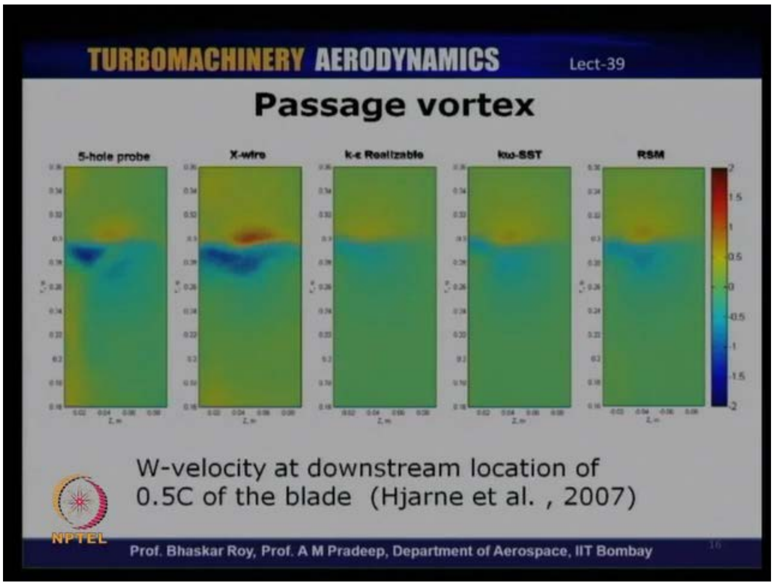
So, what do they find what is plotted here as the streamwise vorticity distribution at a downstream location which is 50 percent downstream of the blade, this was published in the 2007 TURBO EXPO conference which was held in Montreal in Canada.

So, there are five different plots I have shown here. We have the measurements the first two are the measurements from the testing using a five hole pressure probe and a cross wire which is a hot wire anemometer and then we have results from three different turbulence model the K-epsilon, the K-omega SST and the Reynolds Stress Model.

So, this is what we are looking at is the passage between two different blades and this is the wake of the blade. So, this dark blue region that you see here is essentially the vorticity, because of the boundary layer trend from the blade. So, you can see that the vortex associated with the blade is indicated here. This is the passage vortex and the vorticity associated with the blade is shown here in red. So, this is the vorticity of the blade and passage vortex also predicted by the cross wire well measured by the cross wire hot wire anemometer.

If you look at the three different turbulence models, they predict this in different scales. For example, K-epsilon does not really predict the strength very high does predicts the certain amount of vorticity at the in both these location, but not in terms of magnitude. SST K-omega is better it predicts the passage vortex very well and the interaction between these vortex also has been predicted fairly well. Probably the best of the 3 has been the Reynolds Stress Model where it has been able to predict both these mechanism quite strongly.

(Refer Slide Time: 34:53)

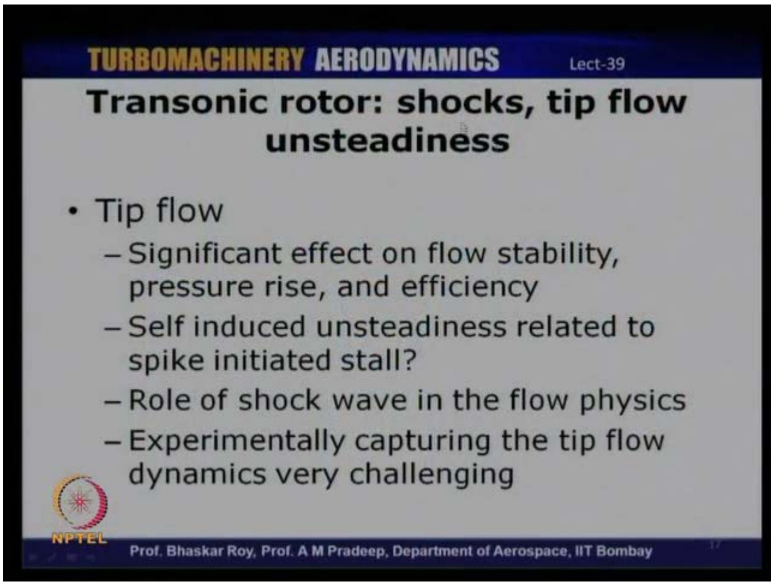


Let us also look at the tangential velocity downstream, at the same location if we measure tangential velocity. Now if you look at the predictions, compare to the experiments, these two are the experiments the 5 hole probe and the 7 hole probe, well the cross wire probe. K-of epsilon model fails to predict it in in terms of magnitude, you can see of course, it does predicts certain amount of w velocity, but not to the extent which one would have expected SST K-omega is better and again the best of the three happens to be the Reynolds Stress Model which which has been able to predict this passage vortex fairly well. So, this is an example of another group of simulations where of course, they have compare three different turbulence models or three of them considered to be good turbulence model, but you have seen that in the K-epsilon model is does not really predict these flow fields a very well whereas, the other two seems to be predicting at fairly well. Reynolds Stress Model being the best of all the three model that they have tested.

So, these are the two different flow flow fields we have looked at where simulations from two different groups who have tried to simulates entirely two different flow fields and they have been able to predict the flow field quite well and compare it with the existing experimental data and that is where the significance of these research findings is that it would sort of provide a designer some guidance towards what kind of turbulence model should be used for a set of applications.

We have seen that depending upon the applications, these turbulence model, will behave differently basically because each of these model is trying to model something which is not a function of the flow fluid, but it is a function of the flow field itself. Therefore, depending upon flow physics, let us say a compressor operating quite away from stall it is operating near design point where the flow is well behaved. Many of these models will be able to predict the flow very well, but as you throttle and bring the compressor towards stall that changes the flow field entirely. So, a model which was working very well in the design condition may not be able to predict the performance of the compressor very close to stall and that is, where the pitfall lies and that is, where the designer needs to put in a lot of caution while using some these turbulence models.

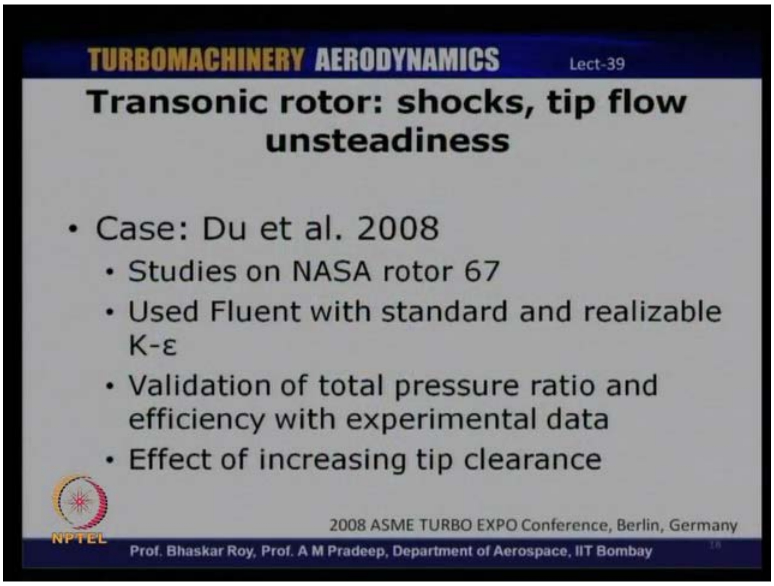
(Refer Slide Time: 37:42)



Let us look at a few more applications, if you look at transonic rotor as we are aware. Transonic rotor is characterized by shocks and tip flow and steadiness and the tip leakage flow as a significant effect on the flows stability pressure rise and efficiency and it been suspected that the self induced unsteadiness is in some sense getting converted to spike initiated stall that we had discussed earlier and the role of shock wave in this entire flow physics is something that is quite challenging, how do you captured this kind of a highly complex flow when you have supersonic flow presence of shocks and over and above that it is a highly three dimensional with high viscous effect associated with boundary layers.

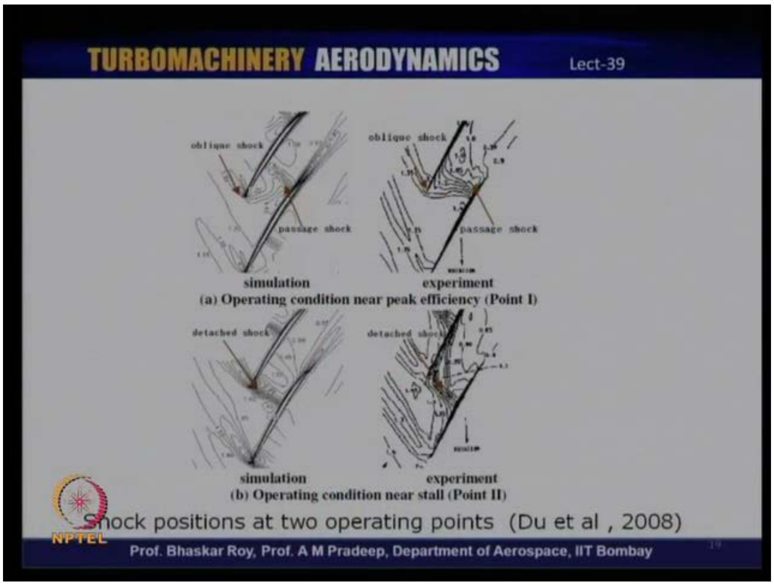
So, this is probably is a very complex flow field as complex as it can get and predicting such a flow is is quite a challenge not just in terms of CFD, but also in terms of experiments because one needs to have experimental data to validate once CFD simulations. So, getting experimental data for such complex fields flow field is also a challenge in itself.

(Refer Slide Time: 39:04)



So, we have one which is again published into the 2008 conference in Berlin, where they are looked at transonic rotor, which is known as the NASA rotor 67, which is an experimental rotor transonic, and they have used a commercial CFD package fluent, I guess, you have heard about it, with standard and realizable K-epsilon model. They have tried validate the total pressure ratio and efficiency with the experimental data, and they have looked at effect of increase in tip clearance on the formats. So, this is one experimental interesting CFD and experimental data combination where they have looked at a transonic rotor with the presence of shocks, and that is the very major challenge both in terms of CFD as well as in terms of experiments.

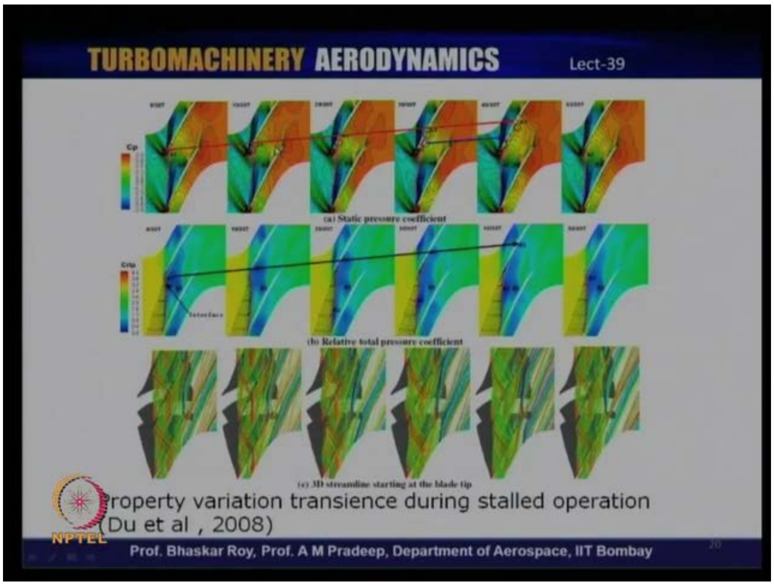
(Refer Slide Time: 40:03)



So, let us take a look at how these results compare. So, if you look at the results, which were obtained from the simulations and experiments; on the left hand side, you have the simulation results, and on the right hand side, we have the experimental results for two different operating conditions: One is near peak efficiency, and the other is near stall. So, near peak efficiency, if you look at there is an oblique shock which as has been predicted by the simulation, experiments also measure the oblique shock at exactly the same locations as you can here. This is the passage shock as you can see, experiments also measure the passage shock at exactly the same location. Now, what happens as you go towards stall, we have detached shock as you can see, experiments also capture the detached shock.

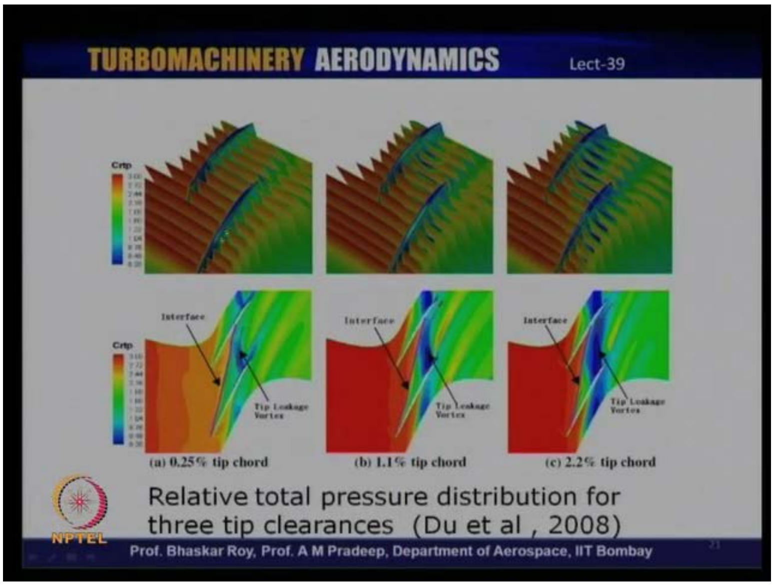
So, this is a set of experimental CFD data combinations, which prediction from the CFD match spot on with experimental data, that too for a very highly complex flow field and using a very commercial software code with the right boundary conditions and turbulence model. So, here they have been able to predict the performance fairly well, and we also have some more results with reference to the total pressure and static pressure, and we will see how that compares with experimental data.

(Refer Slide Time: 41:26)



So, what is being calculated here is the variation in static pressure and total pressure with time, it is a transient simulation, and how the shock position changes as time progresses as you can see, the the variation in the shock position with time is what as shown here especially during stalls. So, this is the stalled operation of the compressor as you take the compressor towards stall, the flow features changed drastically. So, as you can see, this position continuously changes, and by the time it is stalled what was initially here has reached quite downstream of the blade itself. This is also seen in the relative total pressure loss coefficient, which you can see the the blue colored region essentially indicate increase stall, which has occurred; and this is a characteristic of a stalled flow. the 3D streamlines also tell us that the streamlines are all mixed up, and random which indicates the fact that the blade has undergone stall. So, within using commercial codes as well one has been able to predict some of the flow features, which are associated with stall as well.

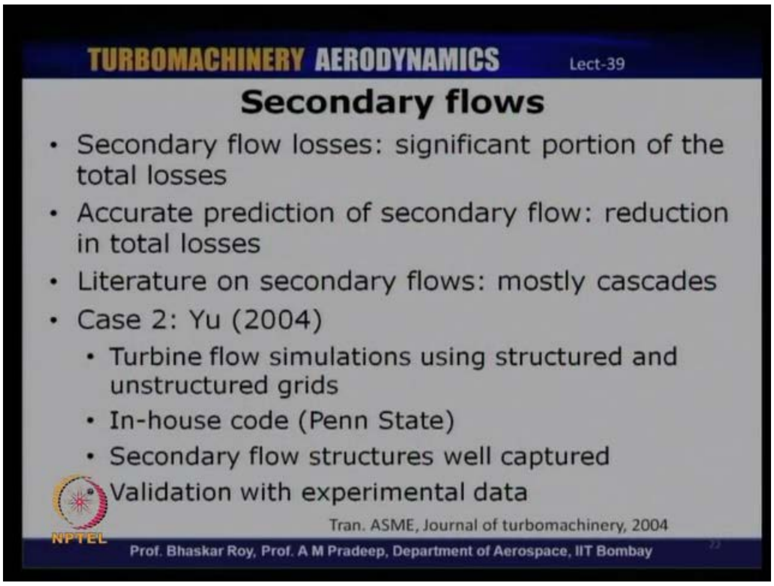
(Refer Slide Time: 43:00)



So, if you will take a closer look at what we had seen in the previous slide, what is seen is that the total pressure distribution that we had seen in this previous slide. If you take a closer look at that, and see what really happens near the tip region what is been predicted is that the blade passage, these are the two different blades, and this is the total pressure contours at different axial locations, and what is seen is that the tip leakage vortex sort of grows as we approach stall and as we increase the clearance from 0.2 percent tip to 1.1 and 2 percent the tip leakage vortex becomes stronger and stronger, which means that that is going to affect the performance of the compressor drastically, because at 0.25 percent chord, the tip leakage vortex as you can see is why quite insignificant, whereas at higher clearance tip leakage vortex in fact, affects the previous blade itself; and that is where it affects, drastically effects the performance of such compressor.

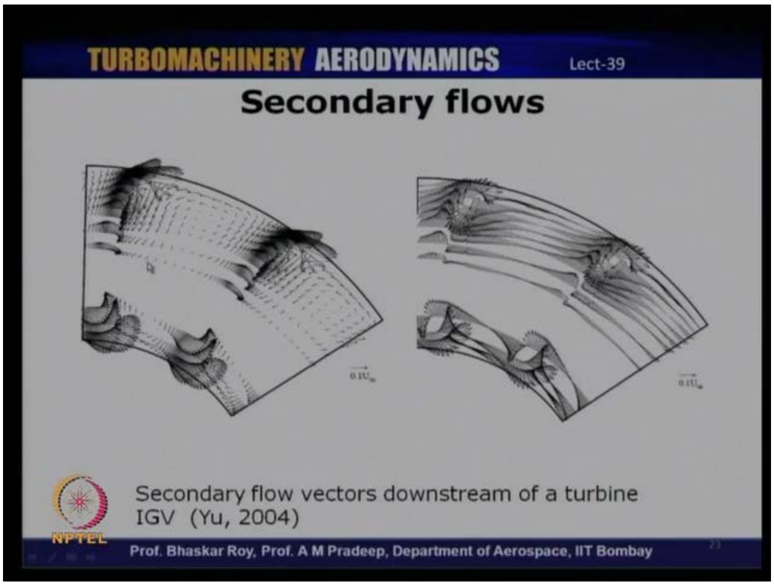
So, we will look at one more case study, which is where they have looked at secondary flows and their effects, and they have been able to predict secondary flows in turbo machine flow. So, this is fairly old study 2004 not quite recent, where they have been able to predict the secondary flows quite accurately in fact, they have used their own in-house code which was developed in Penn state university in the U.S. So, they have used their own in-house code to predict secondary flows.

(Refer Slide Time: 44:29)



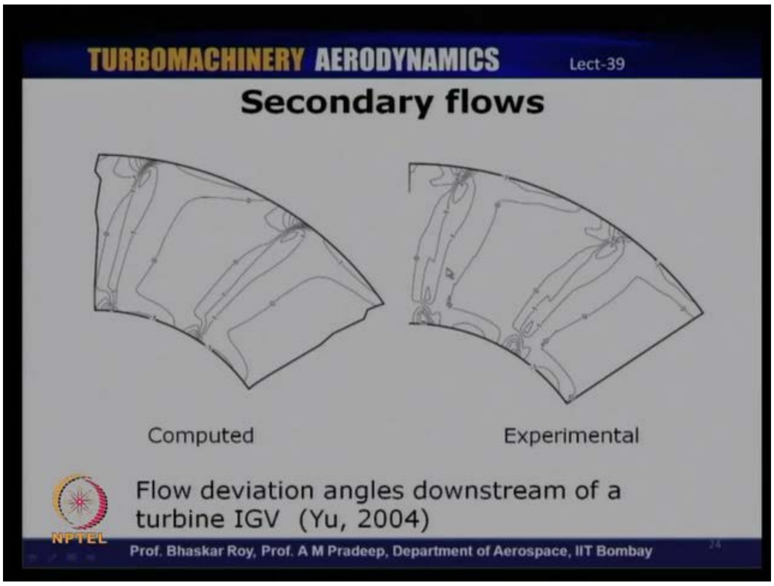
Now secondary flow losses as we have seen is a significant position of the total losses and accurate prediction of secondary flow is very essential, because that can help us in reducing the total losses. So, in this study by you at Penn state where they have used an in-house code, turbine simulations using structure and unstructured grids were used, and they captured secondary flow structures very well, and validated this with experimental data, and these results were published in the journal of turbo machinery in 2004.

(Refer Slide Time: 45:05)



So, here we have the secondary flow vectors, one is the simulation and on left hand side we have the experimental data and this is the prediction from the simulation. So, what you see here is that the secondary flow structure as you can see here as well as that which was captured from the experiment match fairly well. So, you can see secondary flows which were predicted close to the hub as well as the tip and the mid passage, all that has been captured in the experiments as well. So, there is very close matching between what was captured by simulation as well as the experiments.

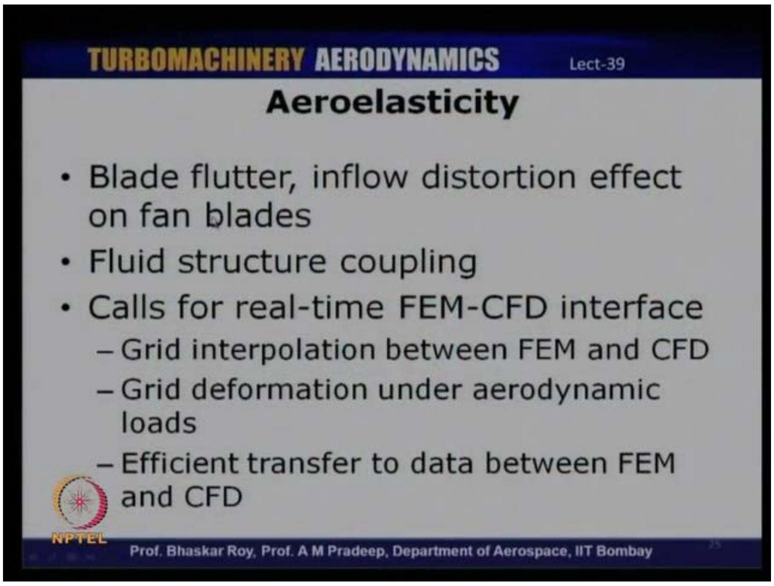
(Refer Slide Time: 45:43)



So, if you look at the flow deviation angles, again from the same steady downstream of the turbine IGV, the deviation angles as computed match exactly with what has been measured from experiments, you can see the deviation angles match quite well. So, both these experiments and simulations match very well, and that is something that gives a lot of confidence to designers in prediction of some of these very complex flow fields, but as I mentioned, there is still lot of way ahead for CFD calculations, we have not matured to a state where we can say that experiments and analytical techniques can be done away with you not need them, that is not the intension as any way CFD is only sub meant to compliment the other analysis techniques like experimental techniques, analytical techniques and CFD obviously, has a lot of potential, but there are certain areas which still need a lot of innovation and research, and one such area is what is known as aero elasticity.

Aero elasticity is something which involves a fluid and the structure coupling, for example, if you consider the fan blades of a large high by-pass ratio, turbo fan engine, the fan blades as you might have seen are huge. So, at design operating conditions the fan blades under and you have seen those blades are very slender as well. So under operating conditions, the blades will sort of under aero dynamic loads will deflect back and forth. So once the blade deflects, there is also a structural effect which will cause a certain amount of vibration, and that will in turn effect the flow and vice versa. So, that is the flow or fluid around the blade has an effect on the structure, the structure has an effect on the fluid and vice versa. So, this is known as the fluid structure interaction and such a problem is essentially refer to as an aero elastic problem and none of the CFD codes are really up to structures and CFD simulations together and that is where a lot of challenge lies in being able to simulate structural aspects as well as fluid dynamic aspects.

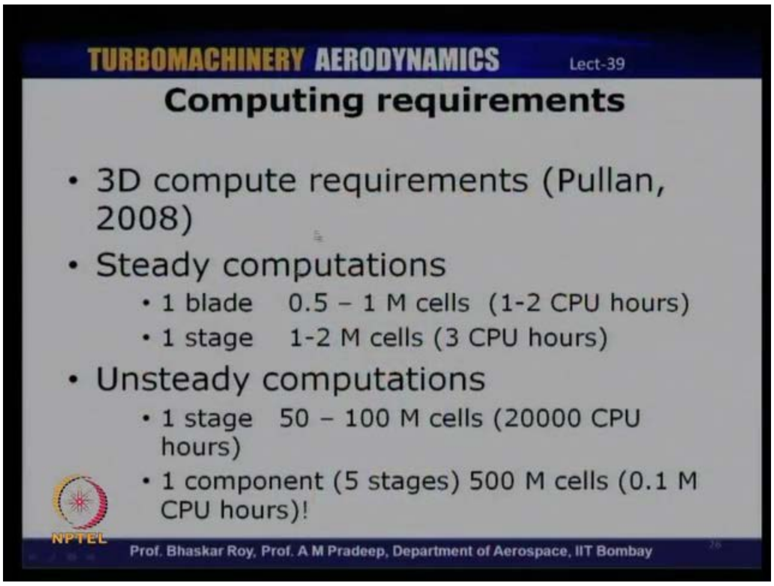
(Refer Slide Time: 48:25)



So blade flutter for example, is one such area which an aero elastic phenomenon, which is also true when there is an inflow distortion on the fan blades. So, there is a fluid structure coupling involved here. This calls for a real-time FEM-CFD interface that is finite element method, which is used by structures people for stimulating properties associated with structural dynamics, and the one which is used for fluid dynamics and that is CFD. So, the major issues here involved are grid interpolation between FEM and CFD, because it is not the same grid that is used, that can be used common for FEM and

CFD, grid deformation under aerodynamic loads and efficient transfer of data between FEM and CFD. So, these are three key key challenges, which are involved in fluid structure coupling. The other major challenge is the computing requirement; so, if you need to do a full 3D analysis take is to next level LES or DNS simulation, the amount of computing power that is required is substantial. So, what is it that we can do to kind of meet these computing requirements.

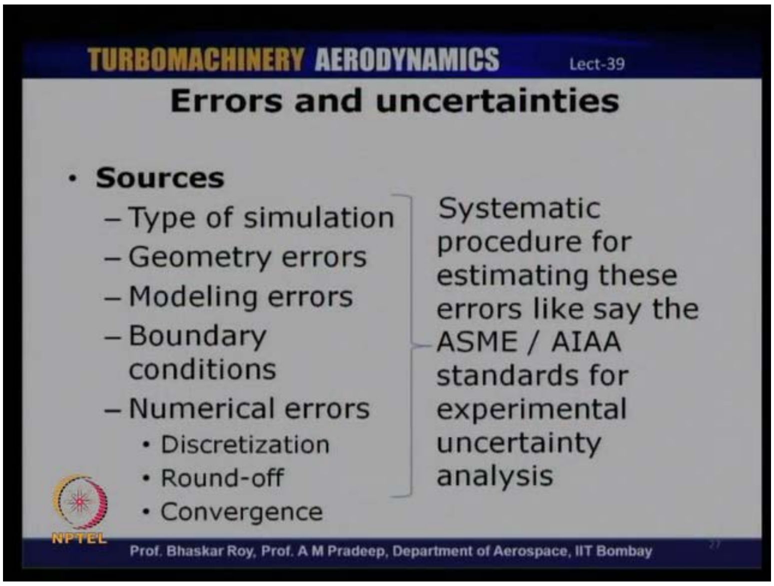
(Refer Slide Time: 49:37)



So, 3D computing requirements as published by Gram Pullan in 2008, these are some predictions, for steady computations single blade stimulation would usually require about point 5 to 1 million cells, and 1 to 2 CPU hours, and one stage which is blade rotor plus stator would probably require 1 to 2 millions cells and about 3 hours of CPU; whereas for unsteady simulation, for one stage you would need something like 50 to 100 million cells and about 20000 CPU hours, and one component of analysis, which is about 5 stages requires about 500 million cells or 1 million, 0.1million CPU hours. So, this is the kind of time that is required for computing power, but the good news here is that the computing power is increasing day by day and we get higher and higher computing power at a cheaper rate, and that is some as saving rates in the sense that we should be able to compute faster and quicker as times progresses and that that way some of this computing requirements should be taken care of. The other major issue that needs to be resolved is the amount of error and uncertainty that is there associated with CFD results;

experimental data we have a well laid out method or procedure for estimating error or uncertainties, a similar thing should also be extended for computational techniques.

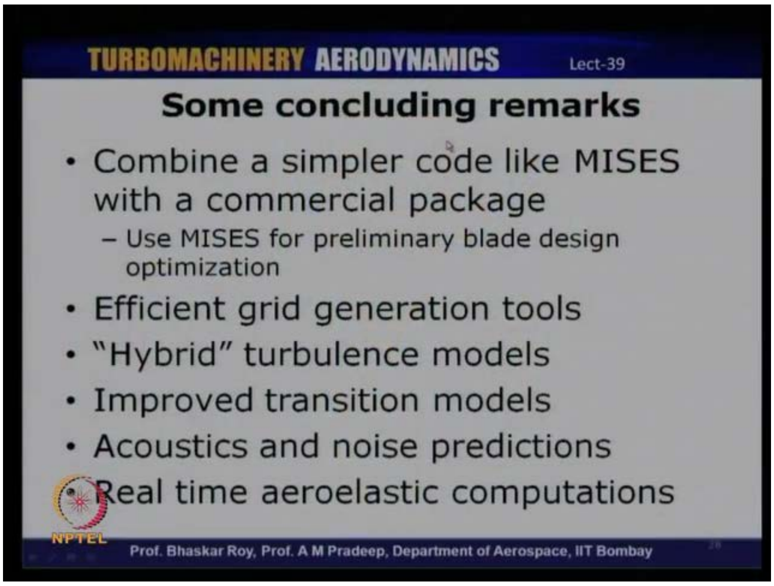
(Refer Slide Time: 51:40)



And what are the sources of errors? Sources of error could be based on types of could be geometry error, could be modeling error, boundary conditions error or it could be numerical error, because of discretization, round-off and convergence. So, given these different types of errors what do we do next. We need to identify as set of procedure systematic procedure for estimating uncertainties and errors. This is true for an experimental analysis, we have standards for like ASME or AIAA standards, which are there for experimental uncertainty analysis, we need to have a certain set of standards, which are true for computational methods or CFD simulations.

So, that is the need of the hour that how do we define or come up with a certain amount of confidence in the CFD data; defining these uncertainty analysis the method of uncertainty analysis should take rough this problem partly. So, let me windup today's with lecture a few concluding remarks, and on what we have discussed in the last couple of lectures on CFD. So, what we have seen is at CFD is a very powerful tool for analysis, at the same time for very complex analysis it can be quite quite computing requirements can can be quite heavy, and therefore there is a need for developing a simpler method which can be used with a certain commercial package.

(Refer Slide Time: 52:46)



For example, one could combine a very simple code let us say like MISES, which is used for simple two dimensional analysis. This can be used for in combination with the commercial package for a preliminary blade design. One needs to still come up with efficient grid generation tool, because that is where most of the time is lost in computation. Hybrid turbulence models which are turbulence models, which can be used depending upon the flow field; I said turbulence models are functions of flow field therefore, can be used turbulence model in different parts of the flow field, which are captured well by certain types of turbulence model. For example, can we use K-epsilon in one part of the flow and can we use Spalart Allmaras in another part of the flow.

So, those are hybrid turbulence models can be come up with such models; improved transition models, aero acoustics and noise predictions and of course, the real time aero elastic computations. These are all key challenges which are something that we need to be worked upon before CFD can be proven to be a an effective tool.

So, that concludes my lecture today, where we had discussed in short some of the challenges which are associated with CFD simulations, and I hope based on some of these discussion we had use should be able to read up more on CFD, if you need to read up basic material on CFD, I would recommend you can read Lakshmi Narayana's book on CFD where he has describe CFD, the basics of CFD in quite detail. you can also refer to some of these general papers and conference papers for more information and updates on the improvement that CFD is undergoing every year, after year. So, hopefully in the years to come CFD would become a very commonly used package, even for very complex flows like turbo machinery applications.

So, I hope you have been able to grasp some of the aspects of CFD through the last two lectures and one of the lectures which will follow this. So, that brings me to the end of this lecture, and this would be my last lecture in this lecture series; and I hope you have had a nice time in understanding some of these aspects of turbo machinery flows. In case you have more questions or queries, you can always get back to us; our contact details have been given to you in the first lecture that we have had, you can always get back to us through email and we should be happy to try to answer your questions. That is it from you, thank you very much.