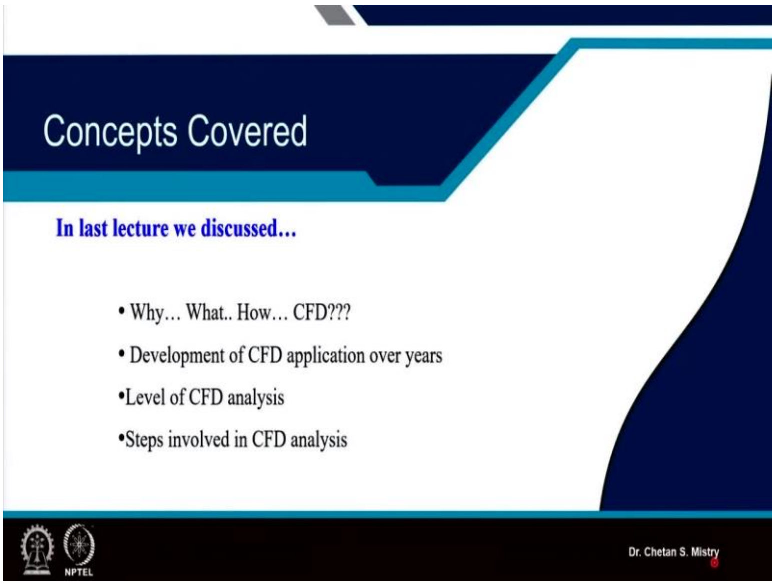
Aerodynamic Design of Axial Flow Compressors & Fans
Professor Chetankumar Sureshbhai Mistry
Department of Aerospace Engineering
Indian Institute of Technology Kharagpur
Lecture 66
CFD Application to Design and Performance Assessment

Hello, and welcome all, lecture 66 - CFD application to design and performance assessment.

(Refer Slide Time: 00:37)



So, in last lecture, we were discussing important aspects about the CFD. The question that was arise in sense, why, what, and how the CFD? Why in the sense why are we looking for doing CFD analysis for compressors and fans? So, basic idea is very first case, it is to be used for design purpose or design assessment. Second, that's what is performance assessment under different design and off design conditions.

Then after, based on that study, if we are looking for something, that question is what, what exactly we are looking for? So, how our flow, that's what is behaving within the blade passage, how the flow that's what is behaving, is it as per our expectation or not? That means, we will be checking with velocity components, flow angles, pressure distribution, total pressure distribution kind of situation.

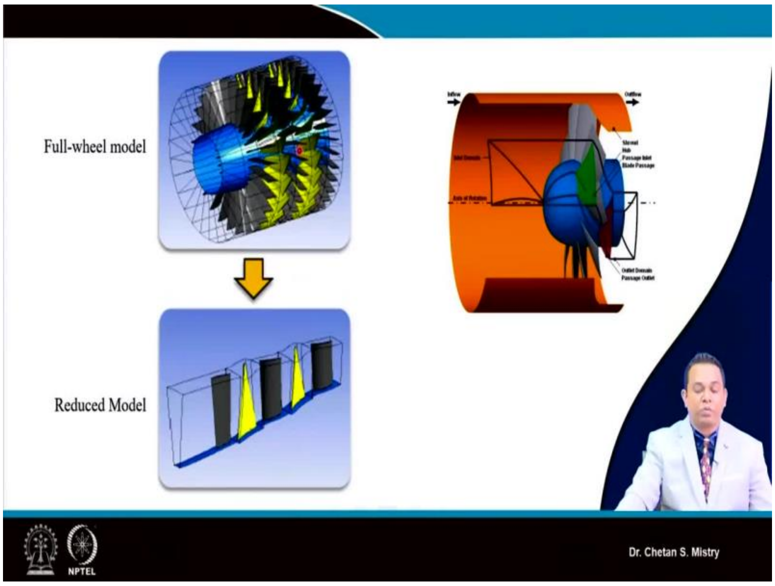
Now, how that means, that's what is giving us answer in terms of analyzing the results what we have achieved by using CFD analysis. So, this is what all we were discussing about. And, in that case, based on my application, all these questions that what will be varying. So, we need to be very careful for exactly which purpose we are doing our computational fluid dynamics analysis.

Then, for turbomachinery application, we have seen the development of CFD for over the year, say initially it was started with 1D, then 2D, quasi 3D, 3D; now, it is full 3D kind of analysis. So, we have seen what all we have learned in that sense. Then after, we have learned with the level of CFD analysis, that's what was initially started with potential flow theory, then after 2D axisymmetric analysis, then it is…it was say quasi 3D simulation, then it is fully 3D in terms we have realized say Navier Stokes equation to be solved; it is RANS, URANS maybe LES or DNS. So, all those things we have discussed in our last lecture.

Then, we started discussing all the steps involved for the CFD analysis, in which we were discussing about initially to make the flow domain then that flow domain we need to divide into small volumes, in order to solve those problems or say in order to solve numerically the flow domain. For that, we are looking for some kind of boundary conditions which are required; thereafter, we are looking for solvers, that's what is using our fundamental equations say mass, momentum and say energy equation.

And, then we have realized after doing this, say analysis, we are looking for post processing. So, from today, we will start discussing about how exactly you will be moving with these steps. So, where is first step, that's what is say, we understand it is to discretize the domain.

(Refer Slide Time: 03:56)



So, let us see what is the meaning of that? Here if we look at, this is what is representing multistage axial flow compressor. Now, we are having rotor, that's what will be followed by stator, rotor and stator combination. Now, if we want to analyze such kind of flow domain, we must realize my volume will be very huge. In order to analyze that we are looking for huge number of elements, number of small volumes that means, it required to have huge computational power, that's what is required huge amount of time in order to solve that problem.

So, we were discussing in place of analyzing say whole stage or a say multi stage configuration, we will be selecting one or two blades, like here it is shown. So, here we are having one rotor, that's what will be followed by stator-rotor-stator combination kind of situation. So, under that condition, that's what will be helping us in terms of reducing the number of elements required.

Now, when we say a number of elements required, that's what will be reducing, that will help in terms of saving our time and computational power. So, in order to do that thing, we need to have certain understanding, we need to have certain terms and conditions to be followed, then only we can analyze such kind of situation. Let me tell, say here, this is what is say we are having stator that will be followed by rotor, then stator-rotor and stator combination.

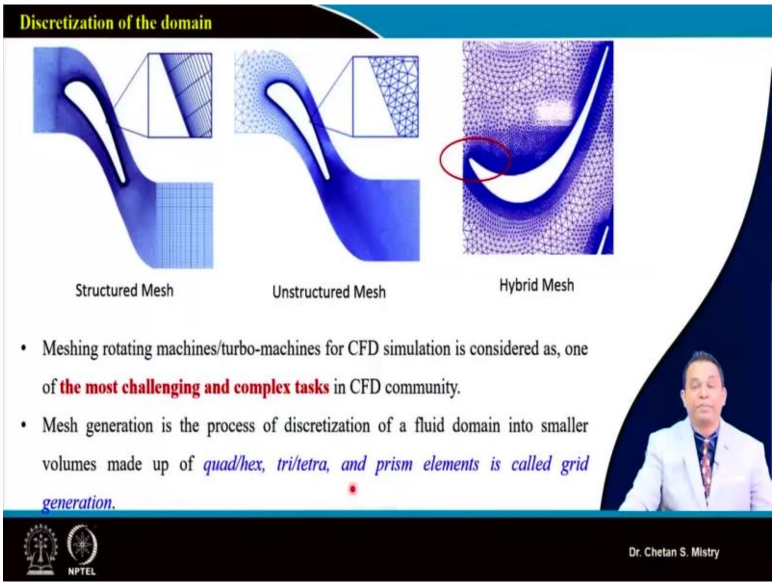
So, under that condition, if you are looking at, this is what is representing one of the domain. Suppose say, this is what is the stator domain, we can say it is a stationary domain. Now, here this blade, it is placed in the center and what surface we are observing on, say, our right hand side and left hand side of this blade, we can say maybe on suction surface and say pressure surface based on the pitch, we need to make this domain ready with.

So, here in this case, what calculation that's what is numerically we are solving that will be getting exchanged to next coming domain. And that is how we are reducing our computational power that's what is called periodic boundary condition. So, here we need to realize one thing, in order to make our flow domain, very first important thing required that's what is to make the blade.

So, up till say week 11 what all we have discussed, that's what is in sense of our preliminary design. Based on preliminary design, we have come up with the blade and blade geometry at different stations. Now, that we are putting in terms of say coordinates maybe x y z or you can convert that into $r-\theta$ plane, $r-\theta-z$ plane. This we will be putting as a input in order to make the solid blade, okay.

Now, once we are ready with our blade, when we are doing our CFD analysis, that blade we need to consider as a hollow portion and remaining portion that's what is of our interest that will be solid. So, just understand one thing, here this blade, what we are talking about, that's what is a hollow passage and remaining flow domain, that's what will be a solid passage and that we will be dividing into number of elements.

(Refer Slide Time: 07:46)



So, let us try to look at what all we need to do here in terms of discretization. So, here if we look at, this is what is representing one of the blade and around this blade, if we look at, we are having systematic arrangement of these elements, that's what we can say structured mesh, we can have unstructured kind of configuration you can see here, this is not systematically been arranged as we can see here. There are some configuration in which we will be having combination of both structure as well as say unstructured mesh, that's what we are defining as a hybrid mesh.

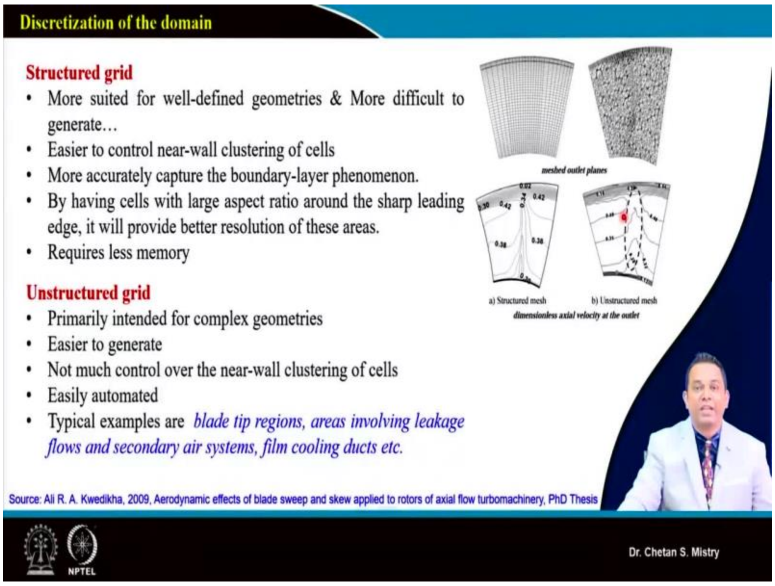
Now, meshing of rotating element in order to do CFD analysis, that's what is most challenging and complex task. Now, when we say we want to analyze say stage…compressor stage, that's what is comprising of rotor and stator; in order to do meshing, one can do meshing within hours, one can take maybe day, maybe very intelligent or maybe very specific person who is planning to analyze the flow domain he may take maybe a week.

Now, the question is what all they are doing? Suppose say randomly without understanding what mesh I will be using, using say commercial package available or say some code that's what is available, straightaway if you are applying, that also will be discretizing our domain and that also will give the solution. But the thing is what exactly we are looking for what is the purpose of doing CFD analysis, that's what is very important.

So, if we consider we know very challenging task for us is to analyze what is happening around my blade, on suction surface as well as on my pressure surface. Same way, what is happening near my hub surface and what is happening near my tip surface or what is happening in a tip region, that's what is of our interest. What is happening between stator and rotor? That is also of our interest.

Now, if we are having this kind of understanding what exactly we would like to analyze based on that we need to do say meshing. Now meshing, that's what we can use say quad mesh, hex mesh, tri or tetra mesh or may be prism kind of elements, that's what will be used for the generation of say grid. We can say mesh, we can say grid, both the things are same in terms of terminology.

(Refer Slide Time: 10:24)



Now, there are two possibilities what we have discussed, that's what is says structured mesh and unstructured mesh or structured grid and un-structured grid. So, we can say structured in the sense, this is what is more suited for well-defined geometry and more difficult to generate. Suppose if we consider we are having say our fan blade or compressor blade, that's what is having three-dimensionality in its shape. Because we are having our blade angles at entry and exit, that's what is varying along the span, and that's what is giving the three-dimensionality to our blade.

Now, in order to mesh this kind of blade or in order to make such kind of domain, yes, this is what is challenging task. It is easier to control near wall clustering of cells, more accurately captures the boundary layer phenomena we will see what is the meaning of that? By having some say cells with largest aspect ratio around the sharp leading edge, it will provide better resolution in that particular area.

So, if we consider when we are using say transonic kind of airfoils in which we are having sharp leading edge, even we are having sharp trailing edge. There, in order to capture the details, it is more important to have such kind of structured configuration and it requires comparatively less memory.

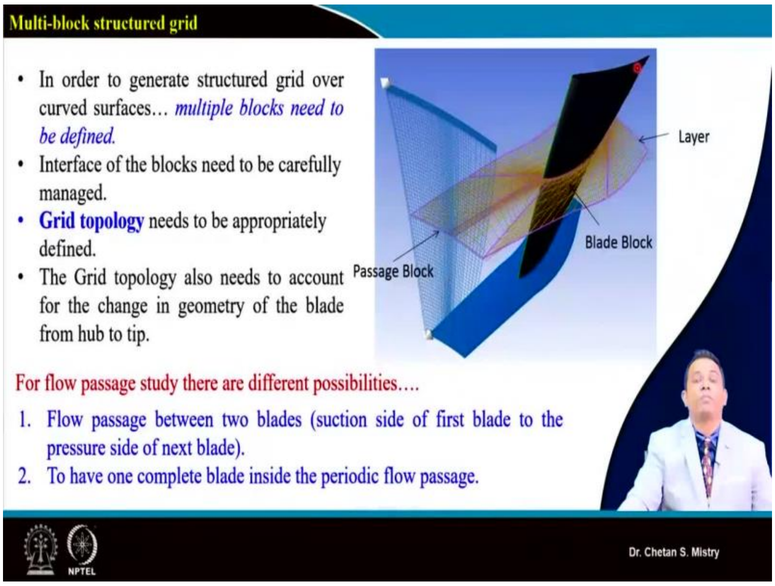
Now, when we are talking about the unstructured grid, it says it is intended for the complex geometries. So, there are some complex flow study what we are doing in order to analyze the flow field for say fan as well as compressor. Under that configuration, we will be going with the unstructured kind of meshing.

It is easier to generate not much control that’s what is possible near the end wall region it is easily automated. Typical flow if you are looking at say near the blade tip region, areas involving the leakage flow or secondary air system, film cooling, the casing treatment, all those kinds of configuration where it is little difficult to do meshing with a small dimension under that configuration we are going with say unstructured mesh.

So, here this is what is a very good example, that’s what is given in one of the theses. Here this is what is representing our structured grid configuration at the exit of stage and this is what is say unstructured mesh kind of configuration. Here if we look at, this whole domain, that’s what is been used with say unstructured kind of configuration and near this tip region, that’s what is a prismatic kind of configuration they have used.

So, this is what is representing say non-dimensional axial velocity contours. So, if we look at here for say structured configuration and unstructured configuration, the difference in a result can be clearly observed. So, now the situation is what kind of flow field you want to analyze, that’s what is giving you idea what needs to be your meshing? That is why I say maybe within few hours also you can mesh, maybe it will take days for doing the mesh, okay. So, this is what we need to understand.

(Refer Slide Time: 13:55)



Now, in order to analyze this part, what we need to do is suppose if I say I am having this is what is one of my blade, there are different methodology that's what has been used by different packages, I say commercial packages as well as say for individual industries, design industries, they are having their own codes, that's what will be used by them in order to generate the mesh.

So, mainly for universities we can say, maybe some professors those who are working with the coding, they may be making code for doing meshing or maybe the students and faculty they need to go with the commercial packages. So, what they are doing? They will be dividing the whole flow field into blocks, okay. So here in this case, this is what is representing the blade block and this is what is representing my passage block.

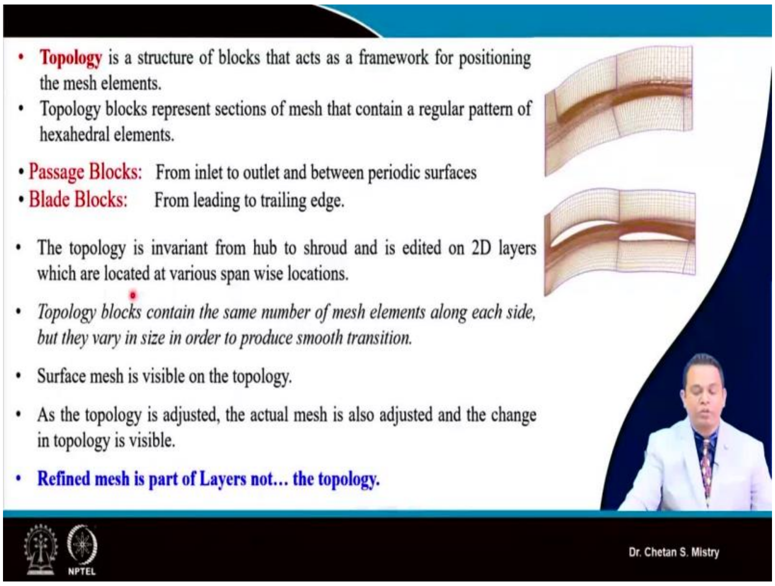
There is one more terminology that's what has been used it is called grid topology; we will see what exactly is the meaning of this grid topology? Now, in order to analyze the flow between two blade passages, there are different methodology people they are opting for.

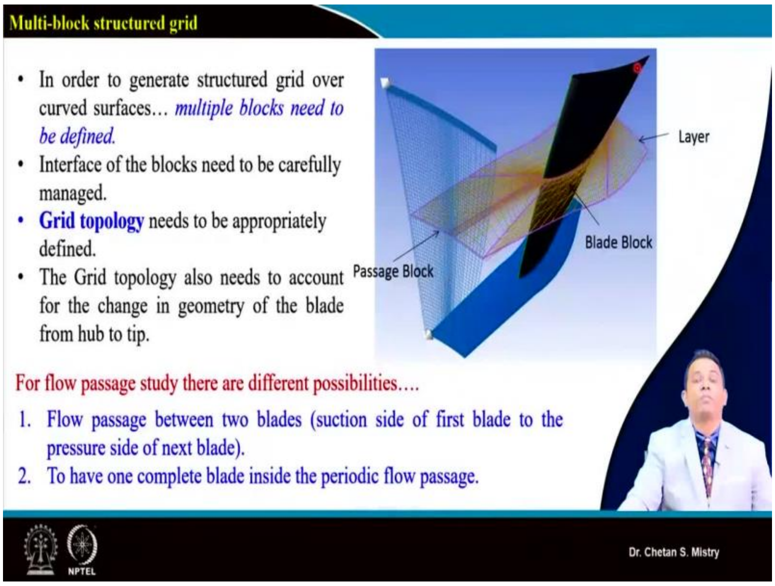
Let us see one of the method, what it says? The flow passage between two blades, say suction side one of one blade to the pressure side of the next blade. So, just realize, suppose say this is what is my blade. Now, in order to analyze this blade, what we can do? We can put our blade at the center and we can have this kind of say pitch configuration. So, half pitch we will be putting here, half pitch we will be putting on other side and this is how we are analyzing the flow.

There is one more possibility in which you can make your blade passage say one of the sides; suppose if you consider, here on the left hand side, you can take your pressure surface or suction surface or on right hand side you can take your pressure surface and suction surface. So, within the blade passage also can be analyzed. So, now, it is your choice how you want to do analysis, this is what is very important.

So, some of the papers when you are referring, they people, they are opting with the different methodologies. So, we need to be very careful about that. Mostly when we are analyzing the rotor and stator configuration, we are opting with this second option, it says complete blade inside the periodic flow passage. So, this is what is my periodic flow passage in which we are putting our blade, okay.

(Refer Slide Time: 16:33)





Now, in order to realize or in order to make this systematic mesh, we need to go with certain topology. So, topology is nothing, here you can understand, this is what is representing my topology at the shroud region, this is what is my topology at say mid-region. Topology in the sense we are making the basic framework. So, here what magenta color you are observing, that’s what is representing my topology kind of configuration.

So, we can say we are looking for doing the mesh or say we need to discretize this domain using these blocks, okay. That block we have defined, that’s what is say passage block and second, we have discussed that’s what is say blade block. So, blade block, it is nothing from that’s what is say from leading edge to the trailing edge and my passage block, that’s what is for say inlet to outlet with the periodic surface.

Now, this topology, that’s what is invariant from hub to shroud and edited on a 2D layer located at various spans. So, you can understand what we have discussed, say; suppose say this is what is my blade. Now, this blade I can divide into number of layers. So, here you can see, this is what is representing one of the layers; like this is what is similar to what we are doing in terms of design.

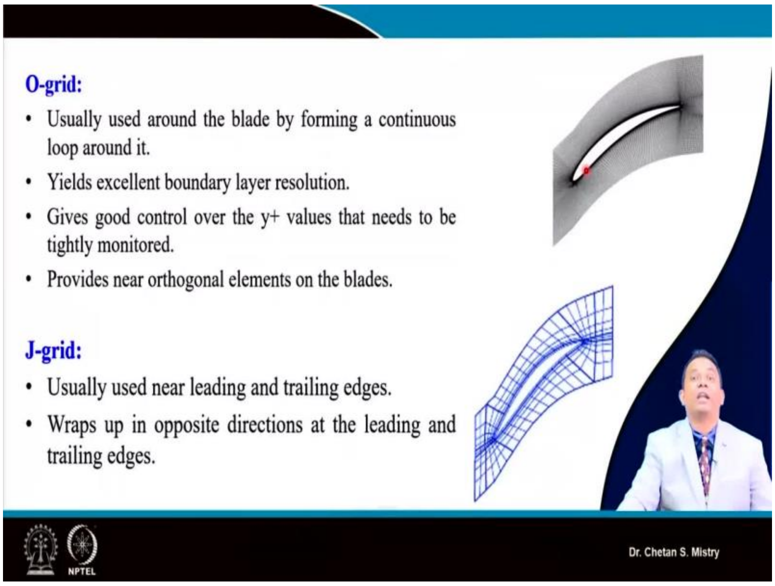
What we have done? We are dividing our span into number of stations. Same way, when we are making our mesh, we will be taking different layers. Now, for that layer we will be making these topologies or these blocks what it will be doing? When we will be doing our meshing, that time it will be taking care of the meshing in a systematic way with increase and decrease of size of these elements, that’s what is a benefit.

So, whatever tool we are using, if we are having a thorough understanding, suppose say this is what is one of the challenging aspects, that’s what is tandem bladed rotor what we were discussing in week 3 also. So, here in this case, my interest is near my leading edge, what flow is happening. Same way, the leading edge of my second blade and the trailing edge or say the exit of my first rotor blade, that’s what is of my interest. Same way, exit also we are looking for in terms of what is happening.

We can understand we want to analyze the effect of change of incidence angle, we are looking for studying the flow physics in terms of wake formation, because of presence of trailing edge, under that condition this is what is very helpful to us. Without blocking also, you can do your meshing, but it is preferred when you are doing the meshing for this axial flow turbomachinery let it be the axial flow compressor, axial flow fan or say axial flow turbine, we will be adopting this kind of configuration, okay.

So, there are some commercial package in which readymade they are making these blocks. With that understanding even using other meshing tools also you can make your say blocks and you can analyze the flow physics. Now, we need to be very careful when we say refinement of mesh is a part of layer. So, we will be doing refinement or maybe we will be changing the size of the element everything that's what is within these blocks. It is nothing to do with the topology, be careful about this part! Now, we can say, once we have made this block or topology what will be our next step?

(Refer Slide Time: 20:23)



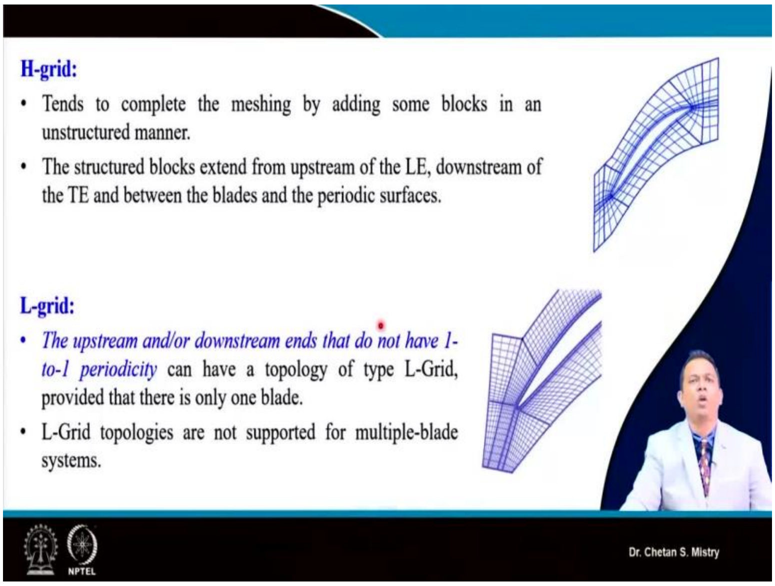
So, now we need to select what kind of grids or mesh elements we will be selecting with. So, here in this case, if you look at carefully, say this is what is our blade. Now, surrounding of this blade, we are having very fine mesh we can observe, that's what is to capture the growth of boundary layer. Just understand what is happening on my suction surface and what is happening on my pressure surface, that's what is of our interest because that's what is representing my work done capacity by individual airfoil.

Suppose if I consider say blade, it is comprising of number of such airfoils. So, for each airfoil, we need to have these details to be available. So, in order to do that part, we need to have great control in terms of y^+. So, you may be aware of y^+, okay, that's what is one of the methods in order to make measurement or in order to make the element near to my blade, okay. So, this is what is representing the y^+ value.

It provides near orthogonal elements on the blades. So, this is what is one of the grid which we can use in order to capture the boundary layer behavior on suction surface as well as on the pressure surface.

Now, next configuration or next kind of grid, that's what is say J-grid. So, here if you look at, this is what is having J shape, what it says? It is usually used near leading edge and trailing edge. It wraps in opposite direction at leading edge and trailing edge. This is what you can say it is making J shape; same way, here if you look at, this is what is making J shape, that's what is say J kind of topology or J kind of grid.

(Refer Slide Time: 22:21)

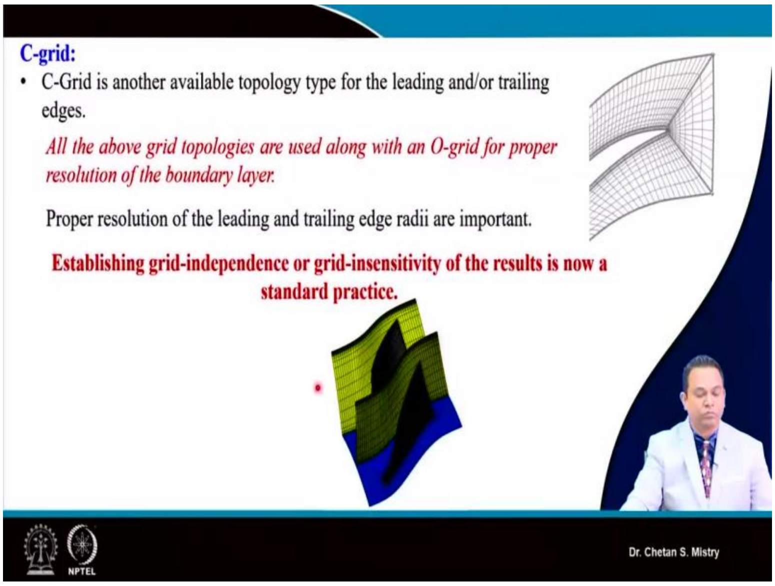


Now, after doing the refinement near end wall region or say near the blade region, now remaining portion that's what needs to be filled with the elements and in order to do that, we will be using say H kind of grid configuration. So, structured block, that's what is extended from upstream to leading edge downstream of the trailing edge and in between the blade and the periodic surface.

So, here if we look at, these all elements, these elements are H grids, okay. Now, in there are say typical kind of blade analysis suppose if we are looking for. So, this is what is one kind of typical configuration we can say. So, upstream and or downstream ends do not have one to one periodicity, here, in this case, this periodicity and this periodicity, that's what is not same, under that configuration, L grid, that's what is being used. It is not being used for multiple blade systems.

So, suppose, when we are analyzing cascades, so, we have seen cascade you can understand that's what is say 2D. But in order to analyze the secondary flow field, we are using that for 3D analysis also. So, for specially, cascade kind of blades, this kind of configuration, that's what has been used that's what is called L grids.

(Refer Slide Time: 23:51)

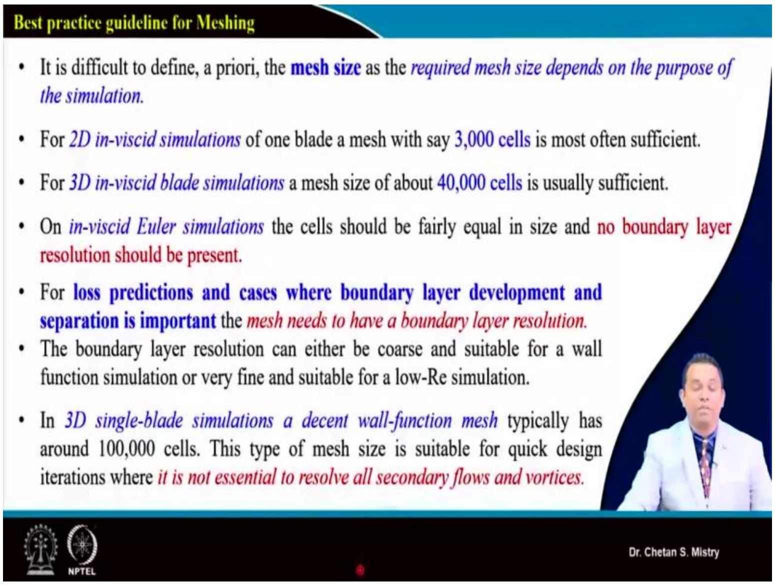


Now, this is what is representing the C-grid, the another way of topology, that's what is provided near the leading edge and trailing edge. So here, this is you can see, this is what is a C kind of configuration, okay. So, all the above topology, that's what has been used along with O-grid with proper resolution in order to capture the boundary layer growth. So, O-grid that's what is very important and that's what we are putting surrounding to our blade.

So, proper resolution of leading edge and trailing edge radius, that's what is very important. So, establishing say grid independence or insensitive, that's what is a standard practice. So, we will be discussing what is the meaning of grid independence study or grid resolution or grid in insensitivity study.

So, here if you look at carefully, this is what is representing the blades and here you can say we have systematic gridding that's what has been done near our say hub surface. Same way, this is what is representing what is happening along the span. So, we can say, this is what is say passage block and near say this is what is of our interest, near the tip region what is happening in order to capture the tip leakage flow we are looking for such kind of elements, okay. And, this is what is representing the periodic surface.

(Refer Slide Time: 25:21)



Now, there are certain thumb rules what people over the years they have defined with and that's what is called say best practice guideline for the meshing. Let us see what we realize for that. So, it is very difficult initially that we will be deciding with what will be the size because we need to understand what is the purpose of simulation as we have started initially only I told the same thing.

We must understand what exactly we want to analyze, okay. So, it says when we are having 2D in-viscous (in-viscid) simulation of one blade, mesh with 3000 cells is most sufficient for. So, you can understand when we are analyzing say potential flow configuration, under that condition we can use 3000 cells, that's what is more than sufficient.

When we are going with the 3D in-viscous (in-viscid) blade simulation, we need to go with say 40,000 cells that's what is sufficient. Now, this is what was earlier analysis. These days also, people they are analyzing using say in-viscous (in-viscid) simulation for getting some of the idea in terms of what is happening with the flow, okay.

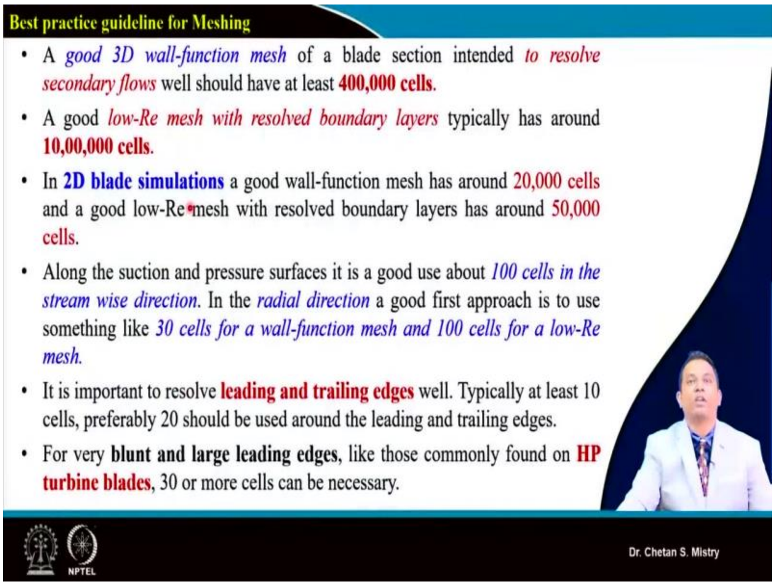
Now, on in-viscous (in-viscid) Euler simulation, it is say it should be fairly equal to the size and no boundary layer resolution, that's what is required. So, we can understand, we are not interested or say it will not give you details what is happening on the surface of my blade, both on suction surface as well as pressure surface. And, that is the reason why we are not bothering of say you having say more number of elements in that particular region.

Now, in order to do say loss prediction, where growth of boundary layer and the separation that's what is very important under that configuration, your boundary layer resolution it is very important, okay. So, boundary layer resolution can either be coarse and suitable for the small wall function, when we are using say low Reynolds numbers.

So, basically when we are analyzing the flow over our blades surface maybe rotor surface or say…our say…stator surface. We are mainly going with the 3D simulation where we are using say wall function mesh there we are looking for say around one lakh cells…one lakh elements. This type of mesh, that's what is suitable for the initial design.

So, suppose say you have made your blade or you have done your preliminary design, you made your blade and you want to analyze that blade quickly, under that configuration small number of cells, that's what is sufficient, but this is what is not giving us idea in terms of secondary flows or the formation of vortices.

(Refer Slide Time: 28:27)



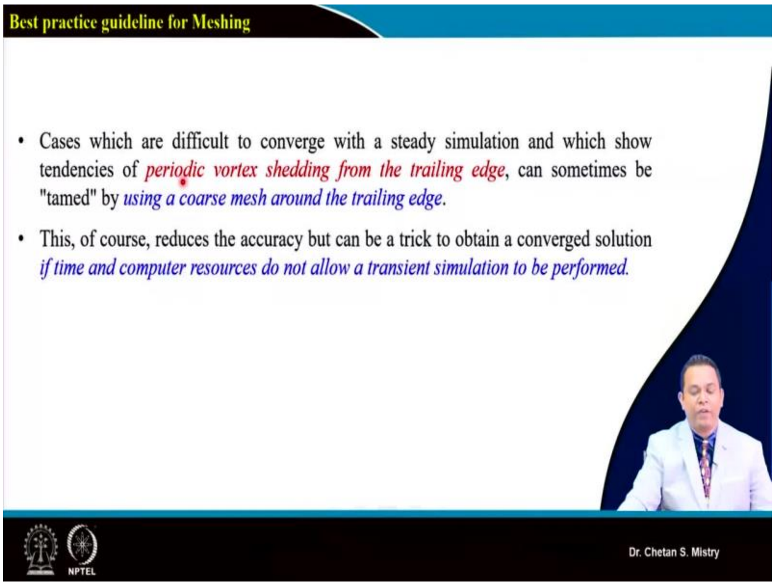
Now, the question is when we are looking for such kind of flow field analysis, when we are going for detail analysis, under that configuration, we need to have the resolution near this blade region. And for that 4,00,000 cells that's what is required. Say, for good low Reynolds number resolution we need to go with say 10,00,000 elements, that's what is say 1 million elements that's what we need to analyze with.

So, 2D simulation also people they are using for initial analyzing for that 20,000 cells it is good for say low Reynolds number mesh, but when we are moving with say analyzing of boundary layer growth, we need to increase the number of elements. Now, all these things what we are discussing that's what will be coming in terms of selecting the number of…a number of elements.

Suppose if we consider we want to analyze the flow in detail near the leading edge and the trailing edge, then we need to use more number of elements in that region. Suppose say we are having blunt kind of configuration, suppose say we are analyzing say turbine, say our gas turbine leading edge, that's what is having say rounded edge; now rounded leading edge under that configuration we need to go with more number of elements.

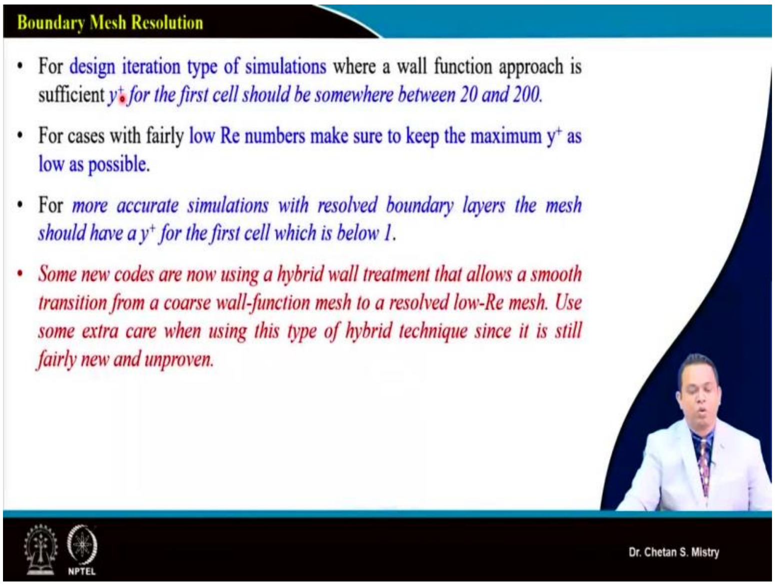
So basically, some thumb rules, some idea that must be there with us when we are analyzing or say when we want to do CFD analysis. Again, the question is how many number of elements what we are using basically it depends on what kind of analysis we are doing, okay.

(Refer Slide Time: 30:06)



Now, it says we are…when we are facing difficulties in sense of say convergence, sometime this…a coarse mesh, that's what has been preferred near the trailing edge. So, this is what is say when we are doing our analysis and we are not getting the solution in a proper way, then sometimes mesh need to be modified. When we are doing this kind of configuration, you know, it will not give what physics what we are looking for, okay. This is what will be used, but this is giving us say low resolution or low accuracy of the results.

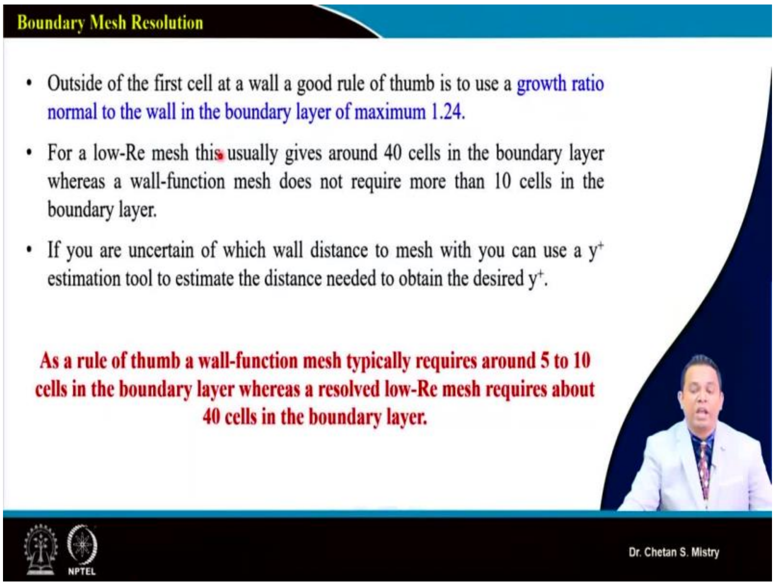
(Refer Slide Time: 30:46)

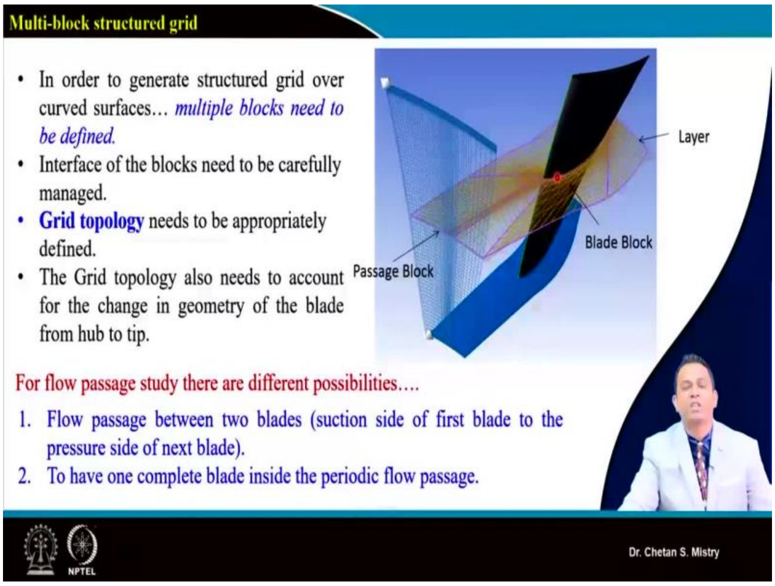


Now, so, design iteration type simulation when we are looking for we need to set with y^+ value. Now, this y^+ value, it says that need to be in the range of 20 to 200 when we are analyzing…we analyzing say low Reynolds number kind of configuration, this maximum y^+ values should be as low as possible.

So, more precise results when we are looking for this y^+ value need to be in the range of 1 or below 1. So, there are some commercial calculators which are available in which you can put your Reynolds number as one of the parameter with some flow properties it will give you the calculation of y^+, this y^+ you will be putting in order to analyze or in order to calculate the value of y^+. That y^+ value, we will be using for making of layers near the blade surface.

(Refer Slide Time: 31:46)

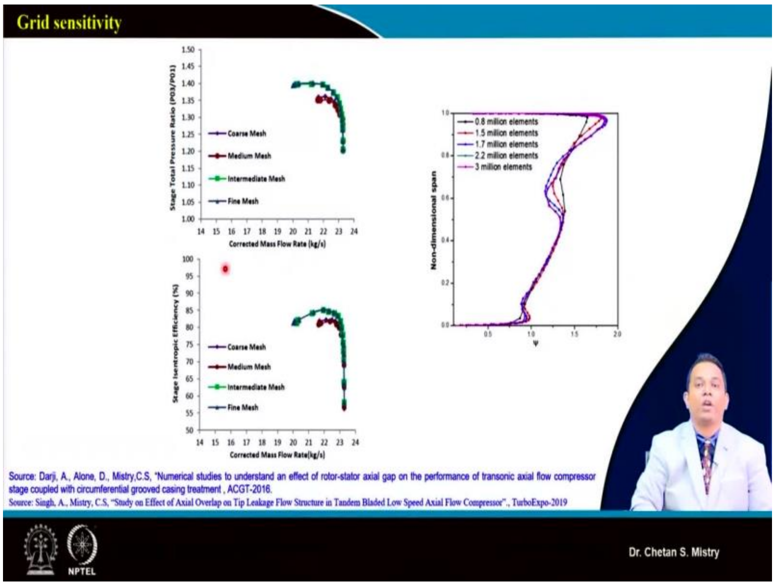




Now, outside say first cell; now, when we have calculated our y^+ value, then we need to move in say direction of periodic surface, that expansion or a growth rate that’s what needs to be kept in the range of 1.24. So, let me show you here, suppose say when we are looking this kind of configuration, here near the blade surface, where we are having say number of elements to be very high and progressively that’s what is going to increase. So that’s what is called growth rate and it says that need to be in the range of 1.24, okay.

So, the rule of thumb, the wall mesh typically required in the range of 5 to 10 when we are having boundary layer to be captured with, when we are looking for say resolve low Reynold number kind of configuration, more number of elements are required near the boundary layer.

(Refer Slide Time: 32:58)



Now, over the years with experience, one will learn all these aspects, we are discussing about say one parameter we have defined say mesh sensitivity. Let us see what exactly is a meaning here. So, in order to check, see number of elements we have discussed sometimes it says maybe 300,000 or 400,000 elements that's what is sufficient, it says more than 10,00,00 elements are required when we are looking for special kinds of analysis. That's what is all depending on the size of my blade.

So, when we are doing this analysis, what elements I have selected that should not change the result. So, for that different researchers, they are opting with the different approaches. So, one of the approach you can see here, this is what is for say, one of the transonic compressor that's what it is designed for pressure ratio of 1.4. Here, under this condition, say initially flow domain that's what has been generated, then for that flow domain, say coarse mesh, medium mesh, intermediate mesh, and fine mesh; 4 different kinds of meshes they have been analyzed.

And, here if you look at, say all this domain, that's what is being run or simulated for different mass flow configuration. And, here if you look at, for say intermediate and fine mesh, we are not having change of results, both in terms of pressure rise as well as in terms of efficiency. So, this is what is called grid independent study.

So, this is what is required whenever we are discussing our results, when we are representing our results in say conference or paper or say maybe talking to industries, that time in order to build the confidence this is what is very important. So very first question, when we are analyzing say initial start with say design, there is nothing to worry for, but when we are doing that systematic analysis, this is what is very important, you need to analyze, that’s what is called grid independent study.

So, you need to go with increasing and decreasing the number of elements. So, here as intermediate and fine mesh, if we configure, suppose say fine mesh, that’s what is required more number of elements rather with less number of elements if you are getting similar kind of trend and result, then we will be going with the intermediate mesh. So, this is what is very important. One more trend, what people they are doing, say along the span they are measuring say total pressure rise.

So, here in this case, this is what is representing say different number of elements which are being selected with. So, particular element when we are selecting, that’s what is not changing my results. So, this is also one of the way in order to select the mesh.

Same way, suppose if we consider, we are looking for say study in which we can take our C_P distribution at mid station, near the hub and near the tip region, for say analysis purpose, we will be doing say mesh sensitivity analysis, or grid independent study at that particular location. And, when we find with increase or decrease of number of elements, with my results are not changing, that’s what we will be selecting for the further study.

So, this is what will give you the confidence to be built. Now, many times what happens? Say, we already have some experimental results available with us. Now, with those results, if you want to analyze, then we need to do the parametric study, that’s what is called validation. So, if I am having, suppose say for this compressor, I will be having my experimental results. So, I will try to match my computational results and experimental results with different mesh and then we can prove that’s what is say independent of grid size. And that’s what will be the helping hand.

So, this particular week, that’s what is dedicated for CFD application to design and performance assessment. So, we need to understand what exactly we mean by design, and what exactly we mean by performance assessment. When we say, again, let me tell you, what CFD we are discussing at this moment, this is not the design tool. This is what is for assessment of design.

So, with this, we are stopping here. Let us see. So, very first aspect for CFD analysis, that’s what is say discretization of domain. So, we have discussed a lot of details in terms of selection of, say elements, type of element, number of elements, grid independence study, that’s what will be giving us more detail idea.

So, many of you are using commercial tools, or many of you are using your already available in house codes just try to read, just try to understand what we mean by all these terms. This course is not for CFD, and that is the reason why we are not going into more detail. You might have done the CFD course maybe in your earlier study or maybe you will be doing that course, that time this will give you more accurate understanding in terms of what we say discretization. So, thank you, thank you very much! See you in the next lecture.