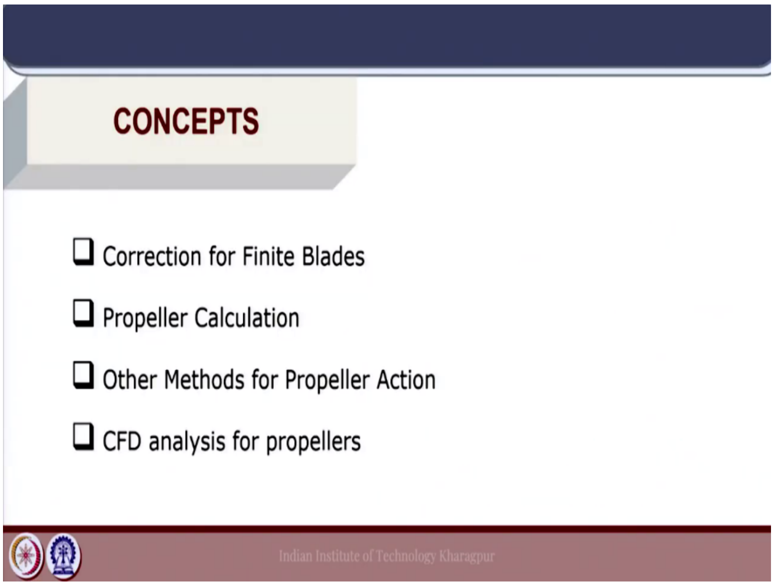
Marine Propulsion
Prof. Anirban Bhattacharyya
Department of Ocean Engineering and Naval Architecture
Indian Institute of Technology, Kharagpur

Lecture - 10
Propeller Theory VII

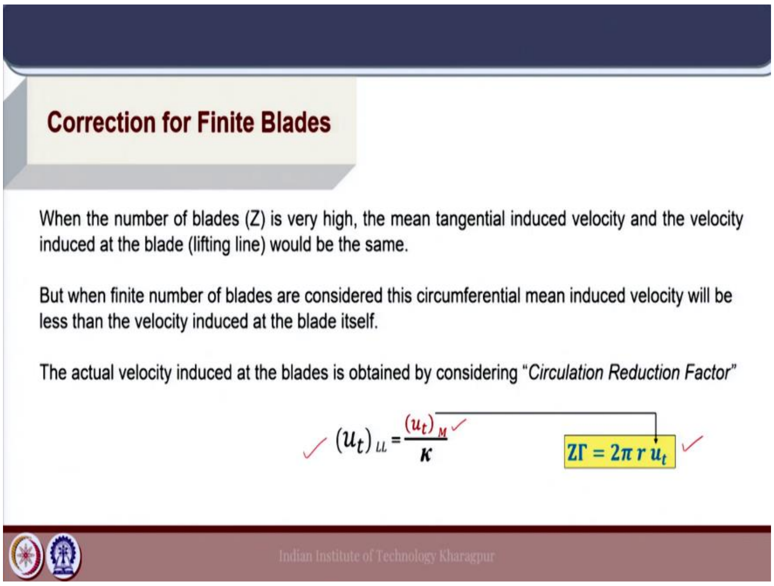
Welcome to lecture 10 of the course Marine Propulsion. This will be the last lecture on Propeller Theory.

(Refer Slide Time: 00:33)



The concepts covered in this lecture will be correction for finite blades, how we calculate the induced velocity from the lifting lines from the circulation theory and apply a correction for finite blades and then use it for a propeller calculation. And then we will discuss other methods for propeller action, lifting surface methods, panel methods and finally, CFD analysis for propellers in brief.

(Refer Slide Time: 01:03)



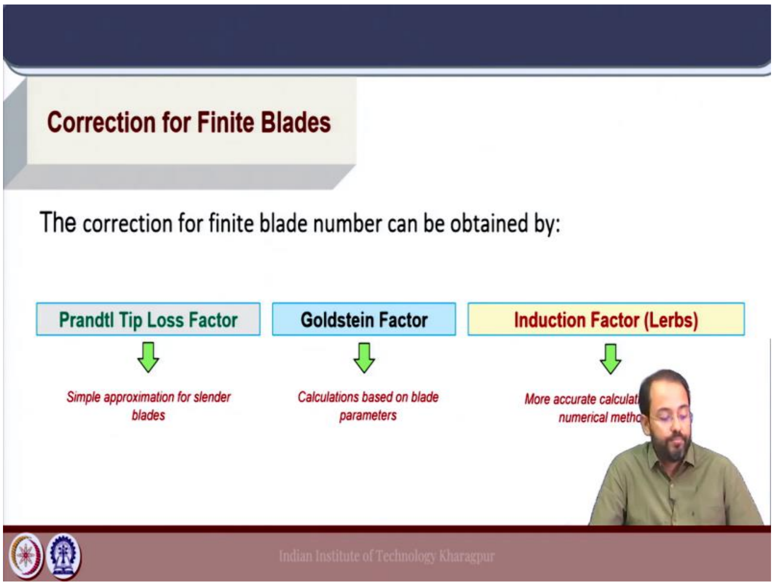
We have observed in the calculation of induced velocity, that the tangential induced velocity the mean of that and the velocity which is induced at the lifting line will be same if the number of blades is very high. But if on the practical side, when we have finite number of blades, the mean velocity cannot be equal to the velocity which is induced at the blade itself. So, it will be low, because each of these blades will shed trailing vortices and they will lead to induced velocities.

But because there is a finite number of blade, the induced velocity will not be uniform in the slip stream and the circumference mean velocity will be lower than the velocity which is induced at the blade. This actual velocity which is induced at the blade now can be obtained using a circulation reduction factor, where we basically apply the correction for finite number of blades.

And we will see some methods that can be applied to relate these induced velocity components. One is the mean one and the other one is the actual which is induced at the blade or the lifting line as we say because, the propeller blades behave like a lifting line. So, this we can write in this manner. The induced velocity u_t the tangential induced velocity at the lifting line will be the mean velocity divided by this factor K which is the reduction factor which we will try to get using some methods.

Now, if we go back to our previous derivation, this one, the ZΓ which is the total circulation for Z number of blades, which is related to the tangential induced velocity u_t, this mean tangential induced velocity $(u_t)_M$, here, is actually the u_t that we have derived in the last class. So, now, we will apply a reduction factor to relate it to the actual velocity that will be induced for finite number of blades.

(Refer Slide Time: 03:25)

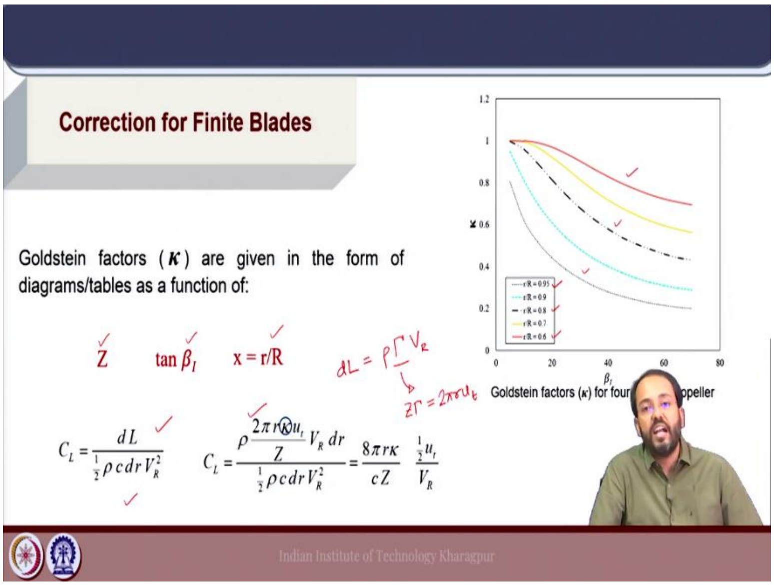


So, the correction for finite blade number can be achieved using different factors. The first one proposed by Prandtl, is a tip loss factor. Another one is the Goldstein factor and the most actual calculation which is more practical in terms of applications is the induction factor by Lerbs. So, the tip loss factor, assumes a very simple approximation of the vortex which is shed from the trailing edges and it is basically applied for cylinder blades.

So, for typically wind turbine blades, which are slender and which shed trailing vortices this tip loss factor can give a relatively good approximation. On the other hand, for marine propellers, we can use Goldstein factors, which is a simple approximation of the relation between the induced velocity which is the mean one and the actual one and these factors depend on certain blade parameters which we will see soon.

And these Goldstein factors give a good approximation of the finite blade effect on the induced velocity. For more detailed calculations, where we use computer generated algorithms, then for higher accuracy induction factor can be used, which has been proposed by Lerbs, where actual calculation of the induced velocity depending on the strength of trailing vortices can be done.

(Refer Slide Time: 04:56)



So, in the Goldstein factor method, we see that the factor which relates the mean tangential velocity to the actual one is dependent on mainly three factors. The first one is the number of blades (Z), the second one is the hydrodynamic inflow angle β_i and the third one is the location of the section on the particular blade. So, Goldstein factors are given as charts or tables, plotted over these different parameters.

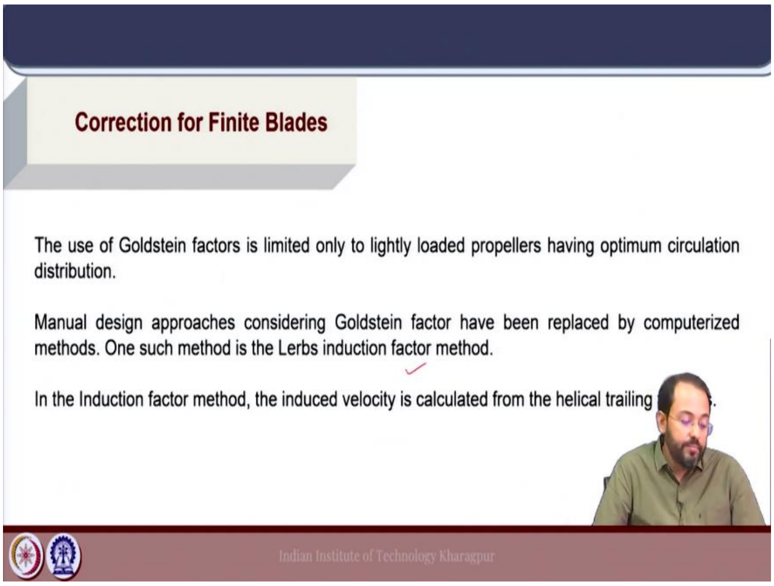
This is a simple graph showing the variation of K, the Goldstein factor with β_i for a four-bladed propeller. We see that at different r/R locations, 0.95, 0.8, 0.6 at different locations, the Goldstein factors are very much different. So, also when we change the number of blades, for example, if we increase the number of blade from 3 to 4 to 5 gradually the value of this Goldstein factor will be closer to 1.

Because, the correction factor will be less as we move from the finite to higher number of blades. So, towards infinite number of blades which was the ideal assumption where we did not use any factor. So, when we reduce gradually the number of blades, the value of this Goldstein factor will decrease. Now, how do we apply this Goldstein factor to the calculation that we obtain using the circulation theory?

We get the sectional lift coefficient C_L using the sectional lift divided by the value $1/2\rho AV^2$, where A is the chord length(c) multiplied by dr. Now, dL is given by $\rho\Gamma V_R$ using Kutta-Joukowski's theorem. Now, this circulation is given by, $Z\Gamma$ is $2\pi r u_t$, which is the tangential induced velocity and hence we can use this to get this expression for a single blade.

The circulation is $(2\pi r u_t)/Z$ and for that u_t, we have multiplied by the Goldstein factor to get the value of the lift coefficient. So, this is how Goldstein factor is introduced in the calculation for the sectional forces that we have done earlier for the circulation theory.

(Refer Slide Time: 08:01)

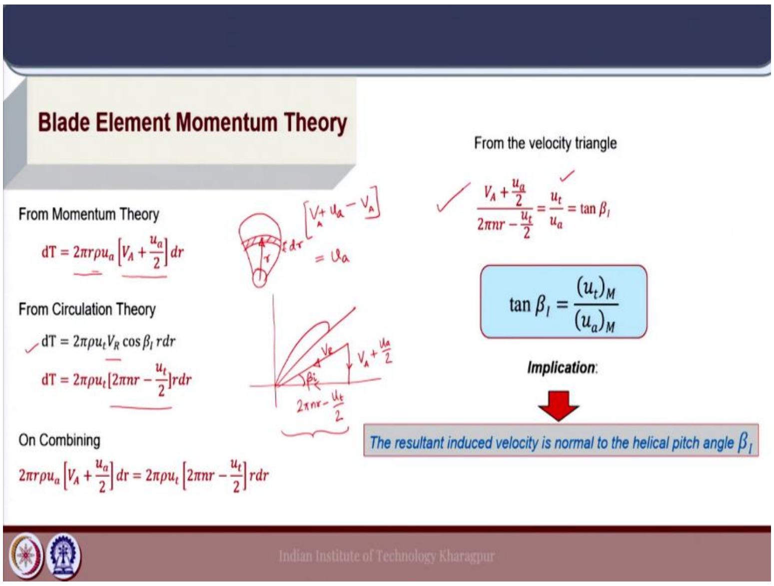


Now, the use of Goldstein factors is limited to lightly loaded propellers, where the distribution of circulation is fairly optimum. But, in more complex cases, where propeller is working in different conditions, where in case of heavily loaded propellers or different design conditions which are not cases where we have optimum circulation then, it is better to use a rigorous method where the induction value is directly calculated using an induction factor.

So, these manual design approaches have been replaced by computerized methods where, the Lerbs induction factor is used. Basically, it uses the induced velocities, which is calculated from the trailing vortex sheets, using Biot-Savart's law or other methods algorithms can be used to calculate the induced velocities from the trailing vortices and that can be actually applied to get the correction factor for finite number of blades.

And then, that can be used to relate the tangential induced velocity, which is at the mean value to the actual value which is generated from the propeller blades.

(Refer Slide Time: 09:19)



Now, another important aspect here is the use of the blade element theory along with the calculation which we get from the momentum theory. Because, often, there are different methods which have been developed for propeller action and the blade element theory gives us the forces on the blade element and also we can use the value from the momentum theory to relate the induced velocity components between the axial and the tangential induced velocity and get the blade forces.

So, the combination of the blade element and momentum theory is often called the Complete Momentum Theory or the Blade Element Momentum Theory, which is used for propeller calculations. So, from the momentum theory, if we apply the momentum theory for a blade element, so, for a particular blade element at a distance r of thickness dr, the volume of water that is passing per unit time multiplied by ρ will give the mass flow rate and that is dependent on the velocity at the propeller disc which is the axial velocity V_A plus half the axial induced velocity.

So, the change in momentum obtained at a particular radius over the entire propeller blade, if this is also based on the infinite blade assumption. So, for the entire propeller blade, the change in momentum will be 2 π r ρ × the velocity at the propeller blade multiplied by the change in velocity between the inflow and a condition which is far astern. So, far astern the velocity is $(V_A + u_a)$ and at the inflow it is V or V_A here, as we have written. So, this will be u_a.

So, the mass flow rate multiplied by u_a, will give the change in momentum between a section far astern and in front of the propeller and this will be equal to the thrust generated by the section at a radius r over the entire propeller blade, ok. Here, I have just drawn one blade to show exactly where we are taking this value. So, we can use this momentum theory to calculate the thrust of a propeller blade section by the change in momentum using the induced velocity in the axial direction.

$$\text{i.e., } dT = 2\pi r\rho u_a \left[V_a + \frac{u_a}{2}\right] dr$$

Similarly, we use the second expression from for thrust from the circulation theory which we have already derived in the last class, where dT is related to the sectional V_R which is the resultant velocity at a particular section and cos β_i, where β_i is the hydrodynamic inflow angle considering the induced velocity.

$$\text{i.e., } dT = 2\pi\rho u_t V_R \cos(\beta_i) r dr$$

Now, V_R cos(β_i) from the blade element diagram is nothing but this value which is 2 π r n -u_t/2 because half of u_t was the tangential induced velocity at the propeller disc. So, if we draw the blade element diagram, with only the final velocity components after taking the induced velocity, this is V_A + u_a/2, 2 π n r - u_t/2. This is V_R is the angle β_i ok. So, the thrust equation can be related to this velocity component, 2 π n r - u_t/2.

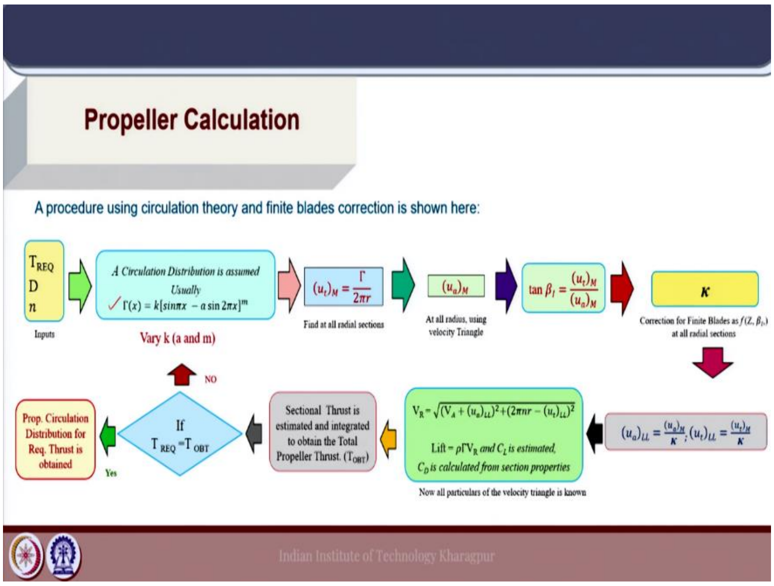
$$\text{i.e., } dT = 2\pi\rho u_t \left[2\pi n r - \frac{u_t}{2}\right] r dr$$

Now, if we combine these two equations, the one that we get from the momentum theory and the other from circulation theory, we get this finally cancelling out the common terms we get this equation. So, this equation gives the ratio of the total axial velocity at the propeller disc $\frac{V_A + \frac{u_a}{2}}{2\pi n r - \frac{u_t}{2}} = \frac{u_t}{u_a} = \tan\beta_i$ as per this velocity diagram.

So, the tangential induced velocity u_t and the axial induced velocity u_a are related by this equation if we use both circulation theory and momentum theory to get the thrust and relate it with the induced velocities. So, we get that tan β_i is u_t/u_a. These are the mean values because we have not applied any correction here. On top of this, we will apply the corrections when we do a circulation theory problem and get the actual values which are induced at the propeller blades.

Because, these are average values for infinite blade assumption where, the number of blades are not considered yet. So, the implication of this particular equation is that, the resultant induced velocity is normal to the helical pitch of angle β_i.

(Refer Slide Time: 15:37)



Now, we will do a simple propeller calculation using circulation theory and finite blade corrections will be applied here. There are different propeller calculations regarding both design and analysis which can be done using propeller circulation theory. We are here presenting a simple calculation where, for a given set of requirement of thrust, we will try to use a circulation distribution and get these induced velocities and match the thrust that we can get for the given circulation distribution.

And this later can be used for propeller design using circulation theory once the circulation that is obtained can be related to the propeller geometry. So, we will be doing a part of the calculation which is relating the circulation distribution to the different induced velocities and getting the thrust from the propeller blade.

So, for the set of requirement of input thrust we define a propeller diameter and rpm and assume a circulation distribution in this form where, Γ is a function of x, where x is nothing but r/R, the non-dimensional radial location on the propeller disc. So, it is assumed in this form, where k, a and m are certain factors, we can start with certain values of this factors depending on the type of circulation distribution that we would like to have on the propeller blade.

So, the factors a, and m will govern the circulation distribution on the propeller blade and we will use this to calculate the induced velocity, the tangential component of the induced velocity which is $\Gamma/ 2\pi r$ for radial sections. First, we will get the mean and then we have to use it later as a correction to get the effect of finite blades for the actual value of the tangential circulation or the tangential induced velocity. The next step will be to get the axial velocity.

In the previous slide, we have seen the axial and the tangential velocity can be linked using the equation shown and we once we calculate the tangential induced velocity, the axial induced velocity can be calculated and from that $\tan\beta_i$ can be calculated using u_t/u_a, the mean values here we are considering. The next step will be to use the correction for finite number of blades basically, the Goldstein factor we can use to convert the mean values to the actual values for the lifting line or the propeller blade.

So, we divide the mean values with the factor which is the correction factor, to get the actual values at the lifting line and this factor will depend on the number of blades β_i and the radial section r/ R which is known to us. The next part, will be to calculate the resultant velocity because, that will lead us to the calculation of the sectional lift force for the thrust and torque.

So, V_R can be calculated using the axial velocity, which is the given requirement and the induced velocities component and the rotational speed we know and also u_t is known now. So, after V_R is calculated, we can have the velocity triangle now. So, all the particulars of the velocity triangle is known.

We can use this to calculate the lift using Kutta-Joukowski's theorem and C_L is estimated from that, the lift coefficient. Using certain sectional characteristics the C_D of the section can also be calculated for particular blade sections that is assumed. Now, from the sectional lift and drag we can next calculate the sectional thrust.

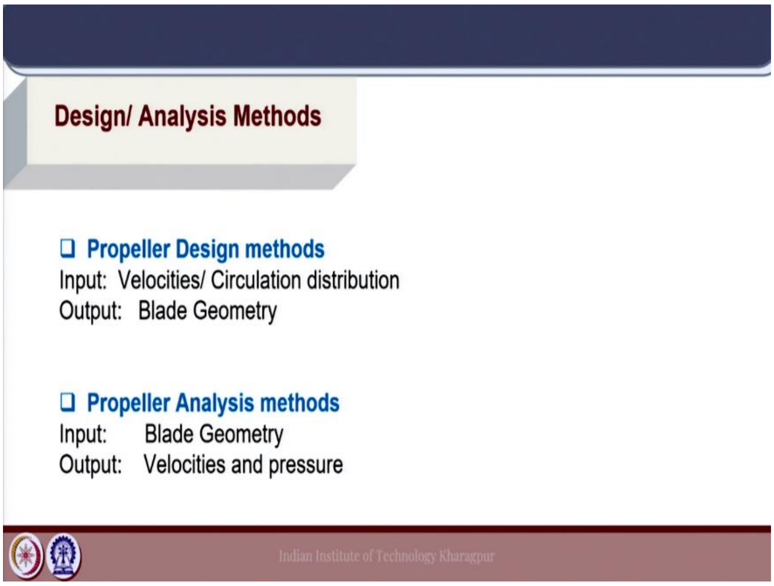
This sectional thrust calculated over different r/R can be integrated to get the total propeller thrust. Now, if this total thrust is not equal to the required thrust. Then, we will go back and change the circulation. Because, this is part of a design process, where we have started with an assumed circulation distribution, this is an iterative process.

So, we have to iterate using circulation distribution values with changed parameters and match the required thrust with the thrust which is obtained after we use the assumed circulation distribution. So, if finally, we get the required thrust is equal to the obtained thrust, then we can go ahead with this circulation distribution and try to define the propeller geometry which is basically part of propeller design process.

So, there are many ways in which propeller circulation theory can be used for propeller calculations. This is a simple example, which is used as a part of a propeller design calculation. Using an assumed circulation distribution, to calculate the induced velocities and then arrive at the velocity diagram and finally, getting the sectional thrust and relating them to the required thrust and finally, which can be used to calculate the geometrical aspects of the propeller.

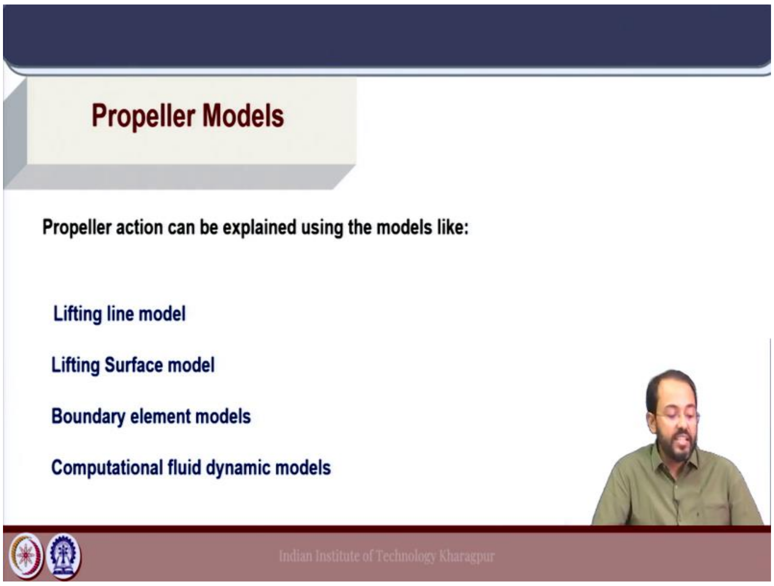
Different variants of these methods can be used for both propeller design as well as analysis.

(Refer Slide Time: 21:38)



Now, if we look into propeller design methods the inputs will be velocities and circulation distribution and the output will be propeller blade geometry. On the other side, if we look into propeller analysis methods, where we already know the propeller design and we want to analyze it, then the input will be propeller geometry and the output will be velocities and pressure and basically the performance characteristics of the propeller, ok.

(Refer Slide Time: 22:11)

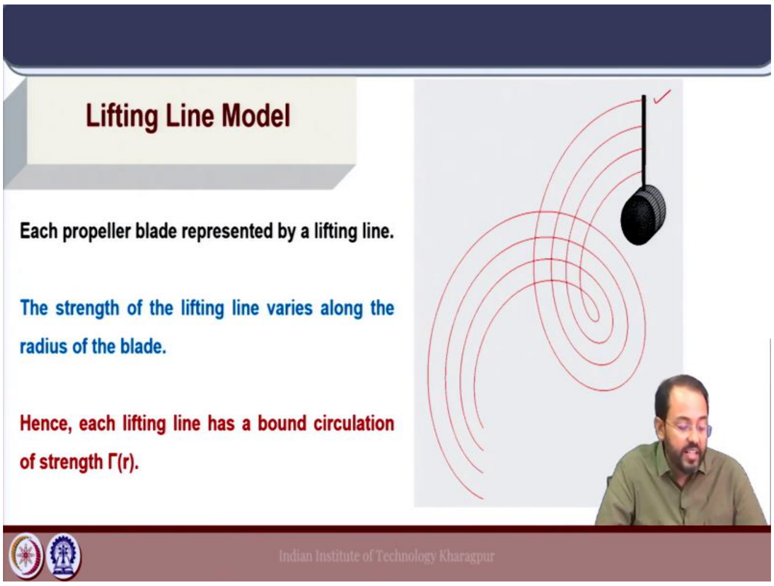


Now, we will move on to other propeller models which can be used for propeller calculations. Propeller lifting line model, propeller lifting surface models, boundary element models or panel methods and finally, computational fluid dynamic models. Now, the circulation theory that we have studied is forming the backdrop for the lifting line and lifting surface models.

So, these models assume the propeller blades as lifting line or a lifting surface with distribution of vortices and that can be used using certain algorithms or in a form of code to calculate the propeller performance. Panel methods are slightly more advanced using variations of thickness and that can be used for propeller calculations.

And finally, computational fluid dynamic models can use the actual propeller configuration and taking into account the viscosity and it can calculate the propeller performance for different operating conditions.

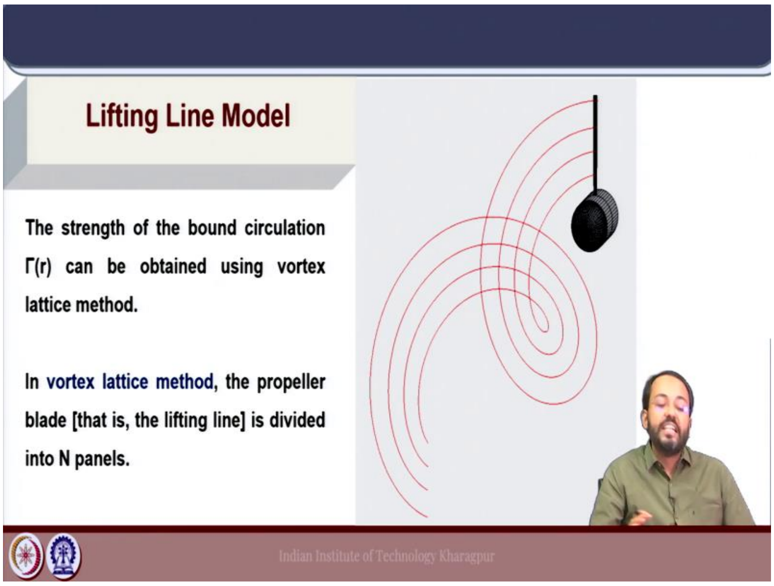
(Refer Slide Time: 23:23)



So, we will briefly go through these models of propeller action. The lifting line model basically assumes the propeller blade to be a lifting line. So, that means, the strength of the lifting line will vary with the radius. Here, the vortex strength is basically given by the bound circulation over the lifting line. Here, the propeller blade is shown as a lifting line and the strength of the bound vortex is dependent only on the radius.

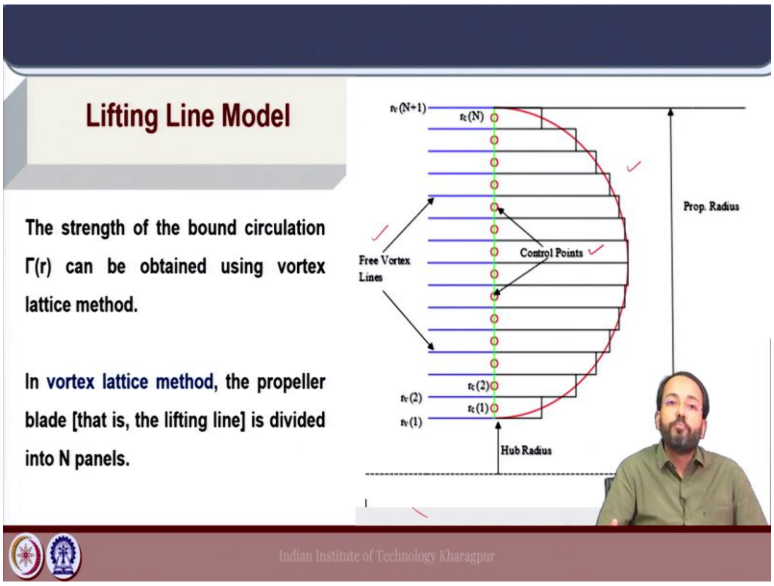
So, because the propeller blade is assumed to be a lifting line, we do not consider the surface. So, Γ is only a function of r.

(Refer Slide Time: 24:01)



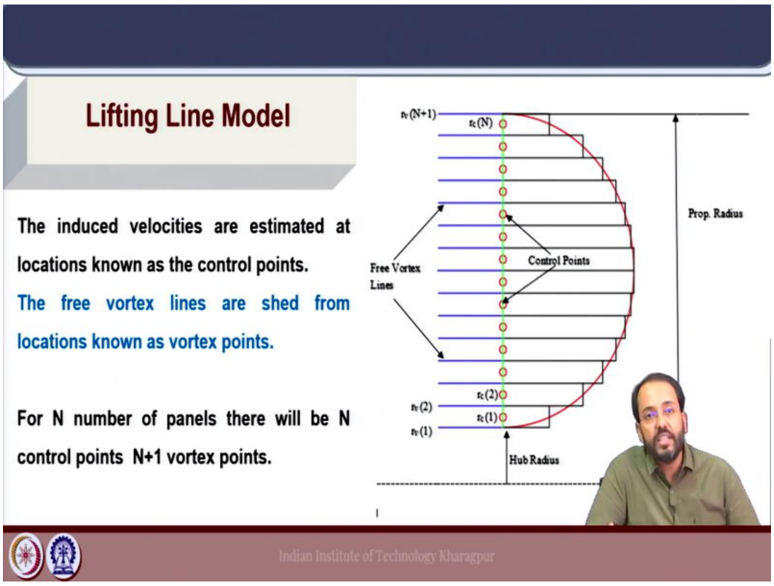
So, the background for this lifting line is the circulation theory that we have discussed. Here, the strength of the bound vortex can be calculated using a vortex lattice method. What is that? We put vortices over the propeller blades in the form of lattice. So, basically the vortices are arranged over the propeller blade and the strength varies in the radial direction. So, in this vortex lattice method, which is under the lifting line model the variation is only in the radial direction.

(Refer Slide Time: 24:38)



And the propeller blade is divided into panels, which is only in the radial direction. The change in circulation is observed. So, these panels in which the propeller blade is divided has a control point at the center of each panel here and these free vortex lines extend from downstream of the lifting line. And the strength of the bound circulation can be obtained using this vortex lattice method depending on the strength of these vortices.

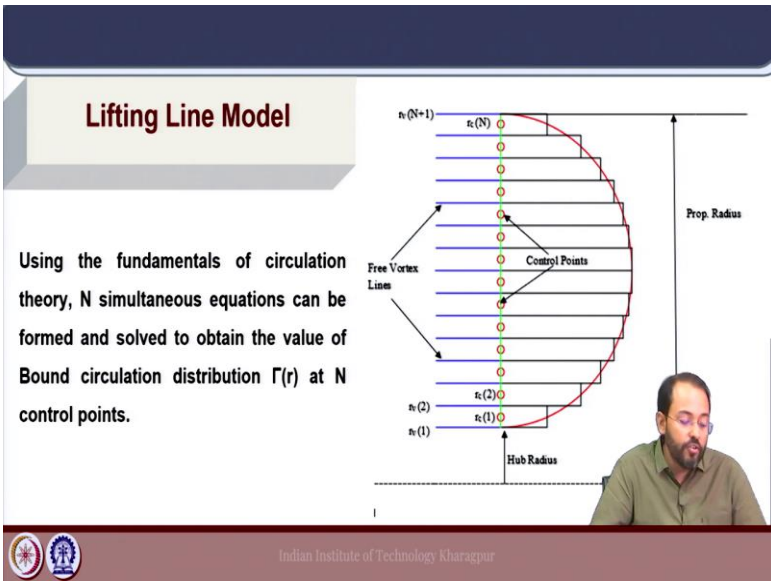
(Refer Slide Time: 25:15)



Now, the propeller blade is divided into panels as shown and the free vortex lines are shed from the locations which are called the vortex points. So, if we have N panels, if we divide the propeller blade into N panels, the number of free vortices shed, which will be from each vortex point will be N + 1, ok because each panel has two vortex lines at the two ends.

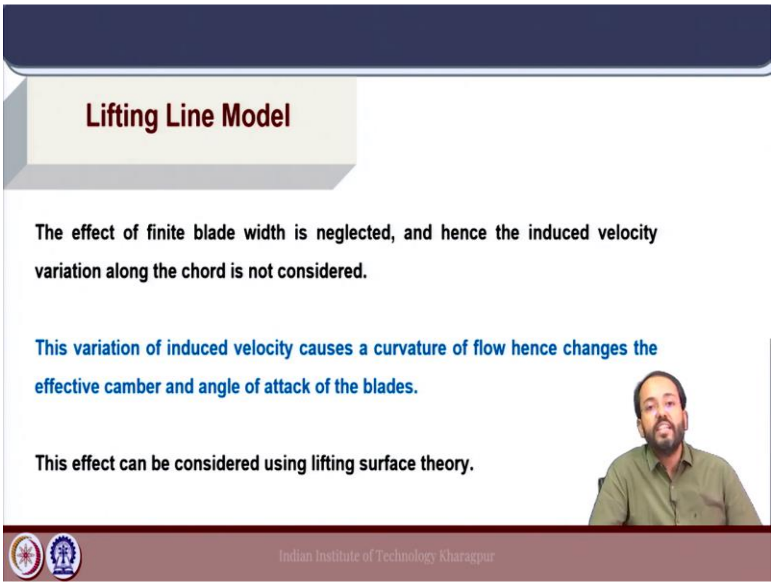
As we have seen previously that, the circulation distribution on the propeller blade changes according to the radius and the more the number of panels the more accurate will be in the form of the circulation distribution over the propeller blade that we get using this lifting line model.

(Refer Slide Time: 26:00)



So, using these values, we can get N simultaneous equations from the circulation value and the bound circulation distribution can be calculated at the N control points.

(Refer Slide Time: 26:13)

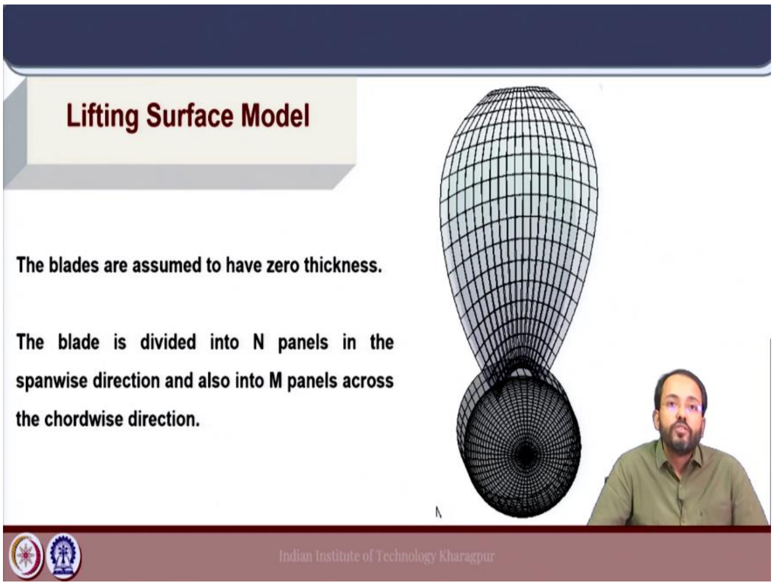


And from this the propeller thrust and torque can be calculated using this lifting line model. And the effect of finite blade width is neglected. So, here we are not taking into care the chord length of the blade. So, we are only considering the variation of circulation in the radial direction.

So, because of that the flow curvature effect and the change of effective camber due to the change in circulation in the circumference direction is neglected. So, that will lead to some differences from the method that we will see later which is the lifti ng surface method where, we will also consider the variation of circulation on the propeller blade in the circumferential direction.

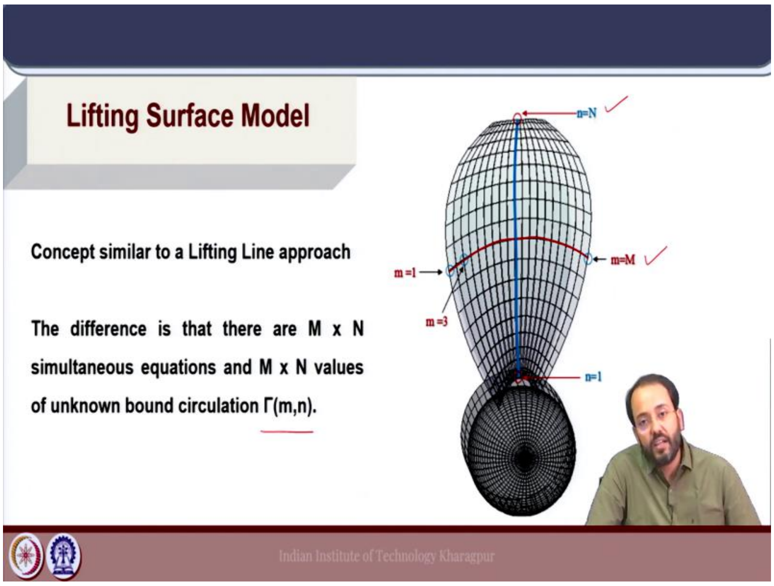
But this lifting line method can give a fairly good approximation of the propeller blade characteristics given the variation of circulation strength in the radial direction.

(Refer Slide Time: 27:13)



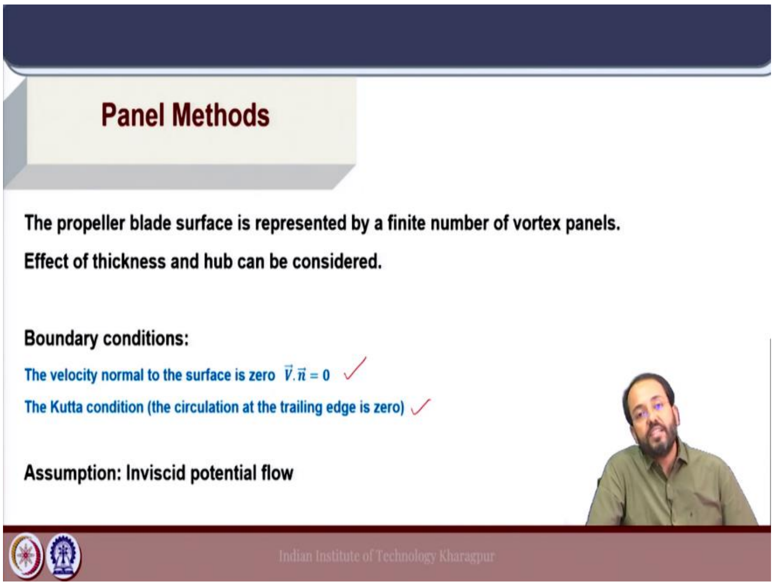
So, when we go to the lifting surface model, here we see that the paneling is done in both the radial as well as circumferential directions.

(Refer Slide Time: 27:20)



So, we have N panels in the radial directions as in the lifting line case. In addition to that, we are also having M number of panels in the circumferential direction. So, N here and M here. For an actual case, the value of circulation will also vary in the chord wise direction. And hence lifting surface model is more accurate in terms of representing the blade circulation as compared to the lifting line model. And different variations of these models with different assumptions are developed to calculate propeller characteristics.

(Refer Slide Time: 27:59)

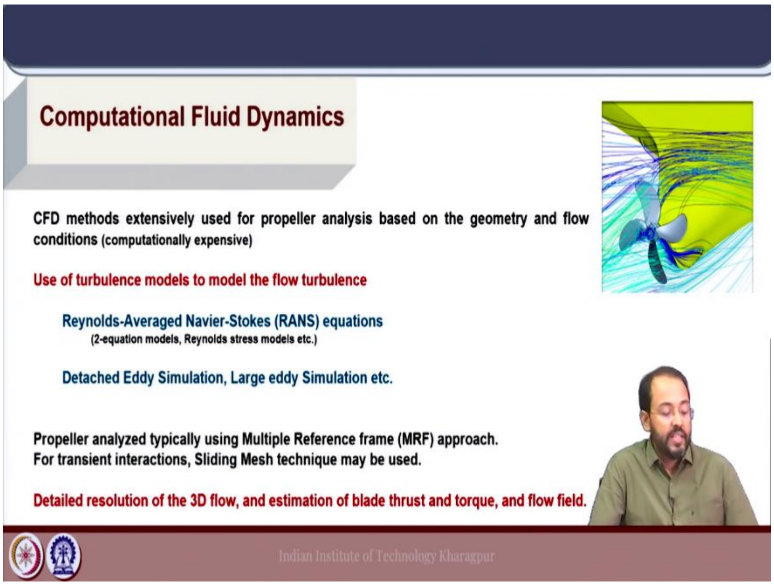


The next set of methods which are popularly used for propeller calculations are panel methods, where by a suitable distribution of vortex panels over the propeller blade surface, the lift generated the propeller blade at different radius can be computed and finally, the propeller performance can be estimated. So, using different boundary element approaches we can have these panel methods estimating propeller characteristics.

The basic boundary conditions that are used are the velocity normal to the surface is 0. And the Kutta condition that we had already discussed which means that the circulation at the trailing edge, for the blade will be 0. These are used as boundary conditions to solve the flow over these panels and get the propeller forces. The assumption is mostly inviscid here using potential flow.

Some effects of viscosity can be taken into consideration using additional models. For more rigorous calculations of the 3D flow around propeller blades, CFD methods are used.

(Refer Slide Time: 29:13)



So, these computational fluid dynamics methods, they are used for both in open water conditions and behind hull conditions in a wide variety of calculations for practical industrial purpose as well as research purposes to develop propeller designs as well as to analyze its performance over a range of conditions.

So, these methods use the propeller geometry as an input and different solvers based on mainly finite volume approaches are used to compute the propeller performance. There are different commercial CFD solvers available in the market and also open source softwares which are used extensively for calculating propeller performance using CFD. The basis of these is the use of turbulence models to model the flow turbulence. Here, we are considering viscous flow.

In the previous models, the inviscid assumption was the basis for the lifting line and lifting surface theories. But, in the CFD approach, we are taking into consideration the viscosity and the effect of flow turbulence is modeled using certain turbulence models. The most popular of that is the RANS method, which is Reynolds-Average Navier-Stokes equations which uses additionally 2-equation models and Reynolds stress models etcetera, to model the turbulence.

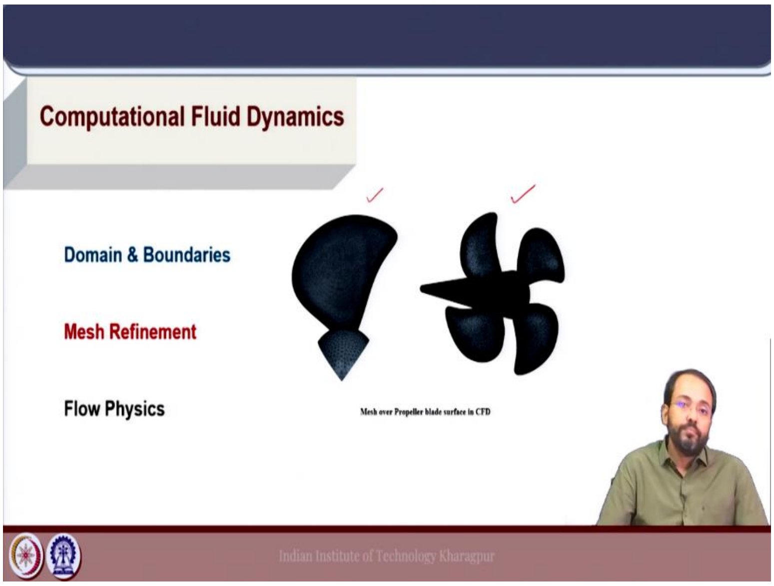
Other advanced methods like, DES, LES, which are Detached Eddy Simulation or Large Eddy Simulation are also used to assess the performance of marine propellers using CFD. In these CFD methods, different reference frame approaches are used to analyze the hydrodynamics of propellers. A very popular approach is to use the multiple reference frame method where, the propeller is working in a rotating reference frame.

And, both steady as well as unsteady assumption can be used in resolving the flow. For transient interactions, additionally Sliding Mesh techniques are used where, different meshes for the propeller region and the outer region are sliding against each other where, proper interface has to be maintained for getting the velocity and pressure values at the different locations of the domain.

So, if we talk about CFD in general, the detailed resolution of the 3-dimensional flow, around the propeller specially the swirling velocity field and the pressure gradients at different locations can be very well estimated if a robust model is used and the propeller thrust, torque and the flow field velocity in the different locations of the propeller blade can be estimated.

Hence, these CFD models are extensively used in propeller design optimization where certain parameters are optimized to obtain proper hydrodynamic performance which is required for specific applications.

(Refer Slide Time: 32:28)



So, if we look into the different aspects of propeller CFD simulations, the essential components are the use of the domain, which is the entire zone in which the propeller is operating. It has to extend to some regions both ahead and behind the propeller and the boundary conditions needs to be satisfied as per requirement of CFD calculations. The mesh on the propeller blade is very important and specific regions on the propeller blade where the velocity and pressure gradients are very high.

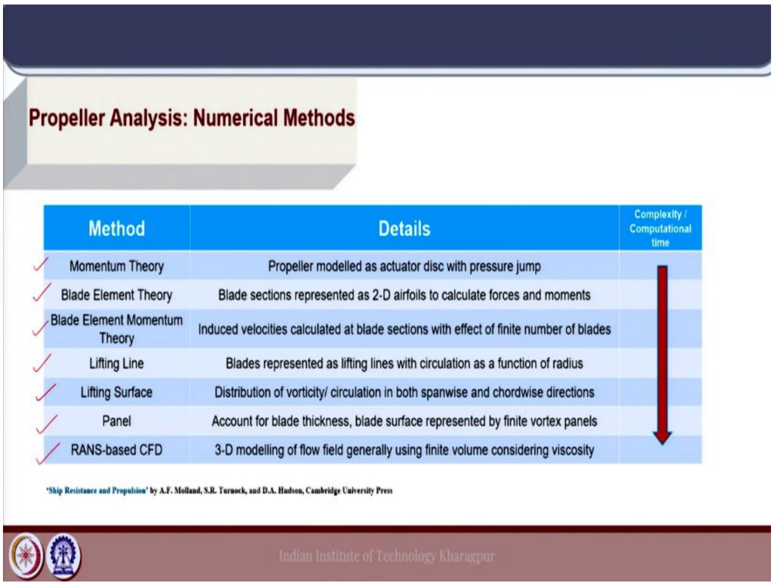
For example, regions close to the blade tip, the mesh refinement needs to be very well defined and on the boundary layer of the propeller blade, sometimes we use special kinds of cells called Prism layers where, the change of the forces typically the velocity gradient over the propeller blade boundary can be well captured because, that is very important for the frictional forces.

There are two setups shown here. In the right-hand side, we have the entire four-bladed propeller and in the left-hand side only one blade for the propeller is shown. So, in a CFD simulation, it is possible to simulate only one blade passage; that means we simulate only one blade of the propeller with periodic boundary conditions to evaluate the hydrodynamic performance of the propeller.

And we can also do a simulation of the entire propeller setup with all the four blades. And depending on the particular requirement, all these cases can be used for propeller performance estimation. And the flow physics which we have already discussed the use of turbulence models, the pressure velocity coupling, these are also very important in the accuracy of the estimation of propeller forces that we get from the CFD computations.

So, in addition to model tests that we perform for propeller analysis, the results from the model test are used to get the propeller performance, in the model scale and they are extrapolated to the full scale. We also use the results of CFD calculations, because this can be done both in the model scale and also directly in the full scale. So, they give a very good picture of the performance characteristics of propellers.

(Refer Slide Time: 35:00)



Method	Details	Complexity / Computational time
Momentum Theory	Propeller modelled as actuator disc with pressure jump	
Blade Element Theory	Blade sections represented as 2-D airfoils to calculate forces and moments	
Blade Element Momentum Theory	Induced velocities calculated at blade sections with effect of finite number of blades	
Lifting Line	Blades represented as lifting lines with circulation as a function of radius	
Lifting Surface	Distribution of vorticity/ circulation in both spanwise and chordwise directions	
Panel	Account for blade thickness, blade surface represented by finite vortex panels	
RANS-based CFD	3-D modelling of flow field generally using finite volume considering viscosity	

Now, if we try to sum up, the different methods that we have used under propeller analysis, this table gives a good representation of all the methods that are typically used for propeller performance. The first one is the momentum theory which estimates the propeller as an actuated disc, which creates a pressure jump and that leads to the propeller thrust. The next one is the blade element theory where we have computed the force and moments over the propeller blade sections.

These are 2D sections at different location, based on the inflow velocity and the rotational speed we have computed the thrust and torque of different elements and integrated it over the propeller blade. Next set of methods which combine the blade element with the momentum theory can help in calculating the induced velocity and find the relation between them which can be used to calculate the forces for the velocities as well as the induced velocity with the effect of finite number of blades.

All taken together we can calculate the thrust and torque for the propeller blade sections and get the final thrust, torque and efficiency of the entire propeller. Next, based on the circulation theory we have, lifting line methods and lifting surface methods. The lifting line method, which represents the propeller blade using lifting lines of circulation varying as a function of radius.

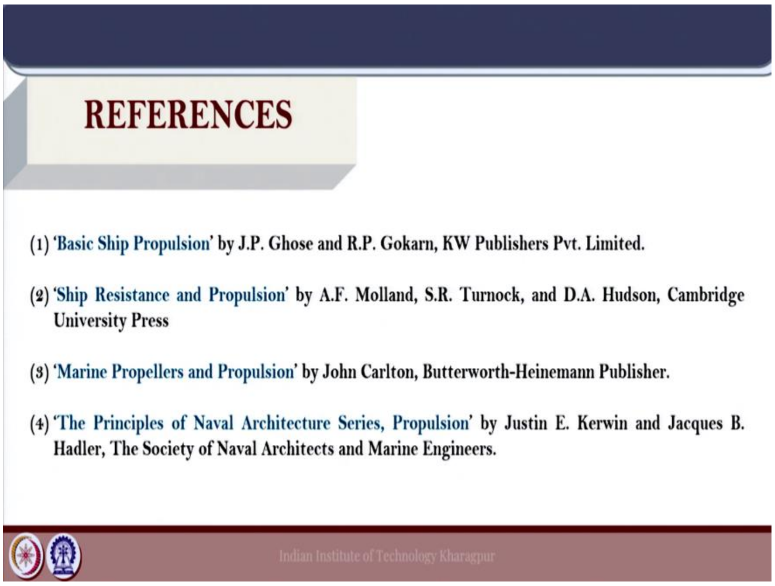
In the lifting surface methods, the circulation varies both in the span wise, as well as in the chord wise direction. We have the panel methods, where the propeller blade is represented by a finite distribution of vortex panels and the propeller performance can be calculated from that and the effect of blade thickness can be included in the panel method.

And finally, we have the CFD methods. The most popular are the RANS based CFD methods which are used for 3D modeling of the flow field for the calculation of propeller forces as well as efficiency, the thrust, torque efficiency both in open water conditions as well it can be as well as applied in the behind hull conditions for self-propulsion cases to compute the propeller performance in a general basis.

So, if we look at the complexity of these methods, the simplest one is the momentum theory and as we go down the table the complexity as well as the computational time increases. In terms of the complexity involved, we can say that the momentum theory is the simplest method and as we go down the table the complexity as well as the computational time increases.

So, we can roughly say that each method is almost an order of magnitude higher in complexity as compared to the method just above it. Now, depending on the actual application or the calculations involved, we can use any of these methods or a combination of these methods for analysis of propeller performance both in terms of propeller analysis as well as propeller design calculation. So, this will be all for the propeller theory part.

(Refer Slide Time: 38:32)



Some references which can be used to understand the different aspects of propeller theory are mentioned here.

Thank you.