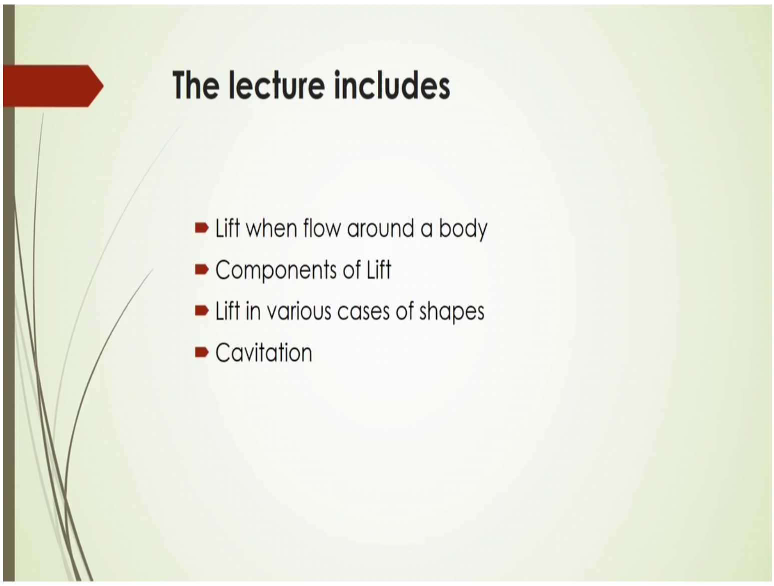
Fluid Flow Operations
Dr. Subrata K. Majumder
Department of Chemical Engineering
Indian Institute of Technology, Guwahati

Module – 08
Lecture – 21
Drag, Lift, Cavitation – Part 2: Lift and Cavitation

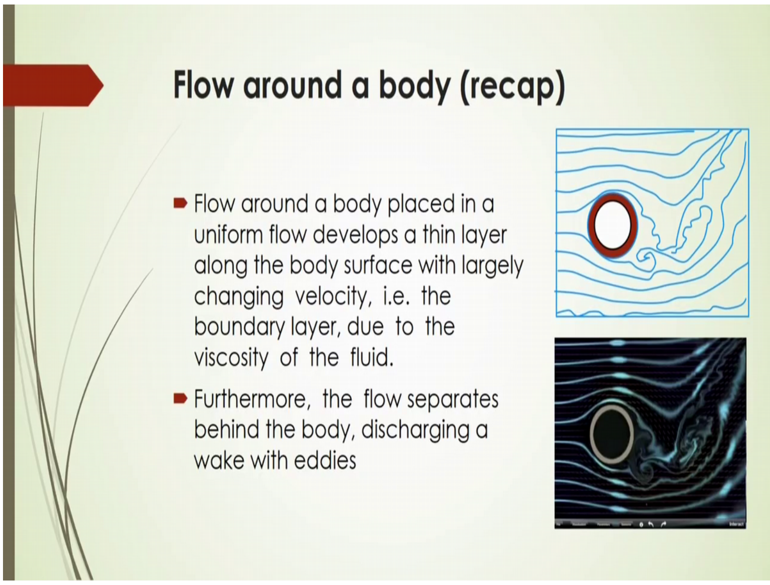
Welcome to massive open online course on Fluid Flow Operations. So, this is the lecture 21 under module 8 as a part 2, the Drag, Lift and Cavitation will be discussed. In the previous lecturer we have discussed regarding the drag and how this drag is conceptually established and also the different variables that effects on this drag coefficients for different shape of particles.

(Refer Slide Time: 01:09)



In this lecture we will discuss about the lift when flow around a body and what are the components of the lift and what are the different effects on lift based on the body shapes and also what will be the cavitation and how it is developed and what are the different aspects of the cavitation will be discussed in the lecture.

(Refer Slide Time: 01:32)



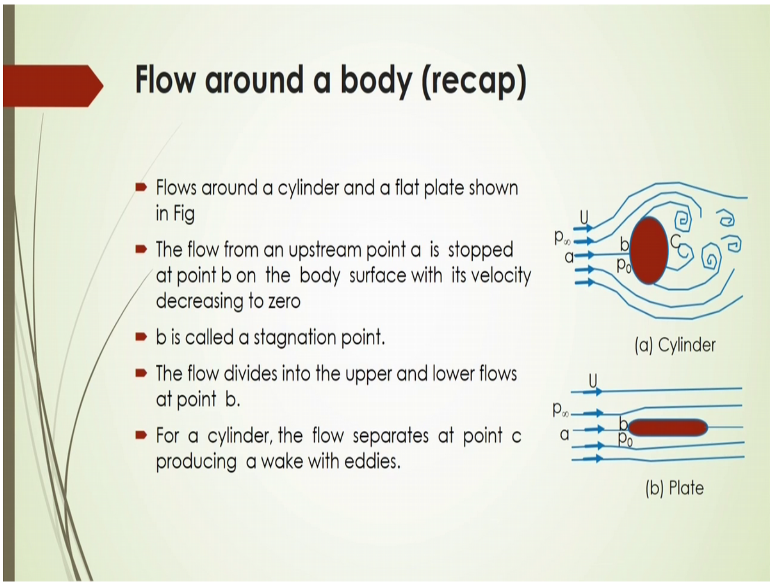
Now, if we consider that flow around a body we have discussed that when a flow is placed in a by uniform flow that is developed in the thin layer along the body surface, in that case you will see there will be a change of velocity because of the viscose effect. And also it is seen that there will be a four separation from the behind of the body and at the beginning you will see whenever the fluid will be strucks at that a stagnant point from that there will be a separation of the fluid into two parts in the upper layer and lower layer of the objects.

And in that case from the upper layer there will be separation of the fluid because of the a drag effect and also viscosity and also other flow regimes effect is there and of course, there will be a shape and that shape depends on the orientation of the object and what exactly it will effect on the drag coefficient as well as other force that acting on the body.

Also we can say that the flow separates behind the body that will discharge a wake with eddies. So, in that case the shape of the eddies would be different for different flow pattern and also it is seen that there will be a formation of the circulation zone behind this cell and that circulation cell will be called as a d and sometimes the shape of this a d would be formed a certain shape of wake and that will change the discharges of the fluid element from the behind of this body.

So, in that case you will see there will be a force acting on that and there will be a some components of the force in the different directions, but we are considering here only two directional components here in the shape whatever the components is coming from the shear force and as well as the pressure force that will be considered here are in this case.

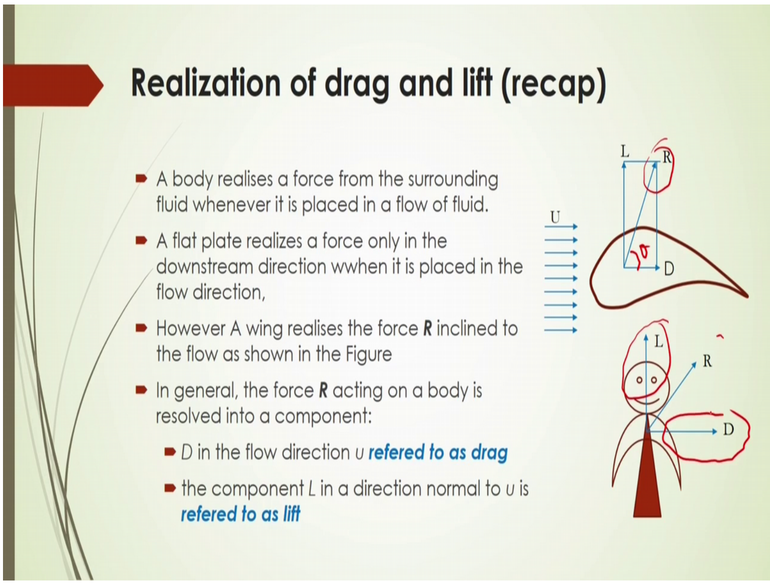
(Refer Slide Time: 03:34)



Now we have already discussed that the flow around a cylinder and a flat plate that shown in figure here in that case the flow from an upstream point a is just stopped at a point b on the body surface with it' is velocity decreasing to 0 here at this position and here in this figure shown and this is called the stagnation point and in this case from this stagnation point from this upper and lower parts of this objects the flow will be divided into two parts.

And for a cylinder you will see the flow separates at a point c by in that case producing a wake with the radius there. So, in that case the shape of that eddies that will formed because of the turbulence and also energy distribution behind this object and purpose there will be a circulation cell and circulation cell whatever it would be form that will be circulates with the opposite direction of this flow compared to each other.

(Refer Slide Time: 04:45)

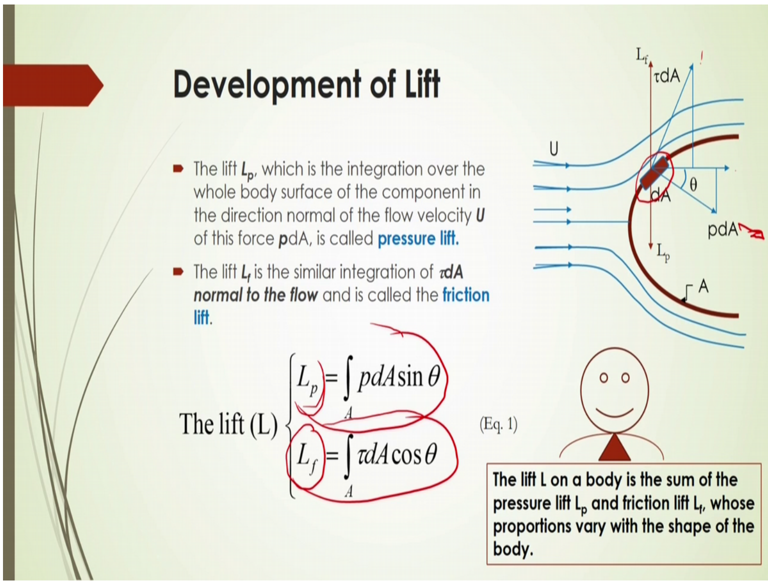


And also you will see that if we considered that how the drag and lift are actually realized or produced conceptually in that case we have already discussed in the previous class that whenever body is a released in a flow. In that case the body will actually acquire a force from the surrounding fluid whenever it is placed in the flow of fluid in the particular direction.

Now in that case for a flat plate it will realizes a force only in the downstream direction when it is placed in the flow direction and; however, a wing realizes the force R as shown in the figure here in this case, this is R this that inclined to the flow at a certain angle that may be at angle theta. So, in that case it will have two components in the flow direction as well as normal to the flow direction.

In general the force are acting on a body is resolve into this components that will be represented by this a drag and also as referred by lift and in that case drag is represented by D whereas, this lift would be considered as a L here. So, in that case D will be in the flow direction U and lift will be in the flow direction normal to the U; U means velocity of the fluid, that we have already discussed how. So, in this direction see here the direction of the lift and direction of the drag is shown here in the picture.

(Refer Slide Time: 06:27)



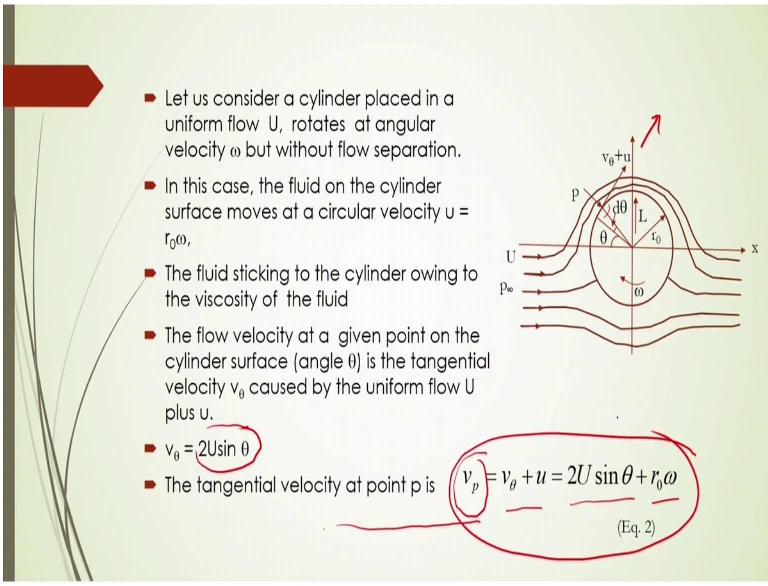
Now, if we consider the lift how it will be developed in that case the lift it will be the integration over the whole body surface of the component in the direction, that will be normal to the flow velocity U. Now, if we considered that in this surface there will be a small area dA and on this area dA there will be force acting in this direction as well as in this direction. Now you will see in this direction the force is acting that will be called as shear stressor if we multiply this is shear stress by it's by area small area then it will be called shearing force.

Whereas that normal direction of this shearing force it will be called as pressure force. Now if we have the components of the pressure force as well as shear force in this flow directions then we will have this p into dA its in this flow direction it will be pdA into cos theta whereas, in this flow direction what should be the component of this shearing force? Itt will be tau dA into sin theta. So, here we can say that the lift L p which is the integration over the whole body surface of the component in the direction that will be normal to the flow velocity U and that will be represented by this L.

Now, this L it will be maybe of that is frictional force or it will be maybe of that is called pressure force. So, two components will some of these total lift force in this case. So, we are getting this normal directions of this flow what will be the flow components that is coming out from the shearing force as well as pressure force are it will be represented as what is that L p that will be is equal to the integration of pdA into sin theta whereas, for shearing force it will be as integration over this area as tau dA into cos theta.

So, we can have this, the total lift force by summing up this two components of this lift that is contributed by pressure as well as the friction this two components. So, you can have this, the lift L on a body it will be the sum of the pressure lift L p and the friction lift L f whose precautions will be varying with the shape of the body. So, we have already discussed these parts, so this is the conceptual part of this how this lift and as well as this shearing force will be developed here.

(Refer Slide Time: 09:15)



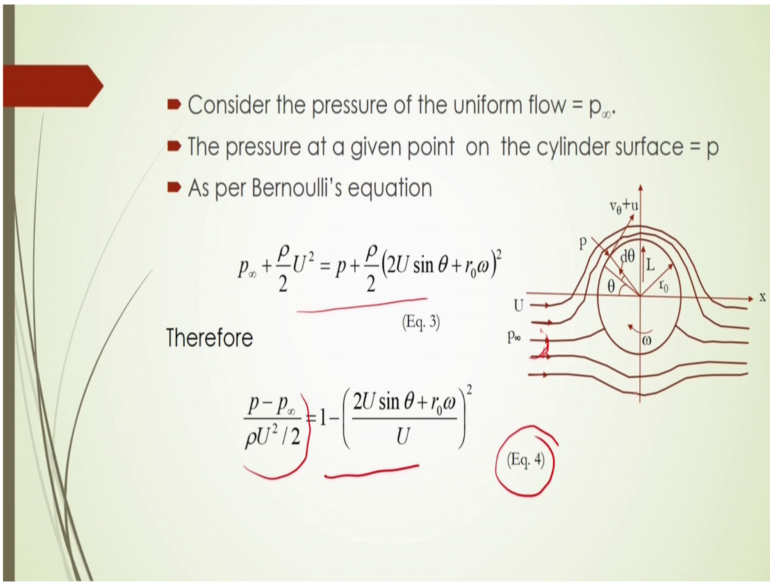
Now, let us consider a cylinder that will be placed in uniform flow U and it will rotates at angular velocity omega, but without flow separation in this case as shown here as shown here in the figure that if it is cylindrical surface and that case flow will be flowing over this cylindrical surface and at the surface you will see this is liquid or fluid will have the angular velocity omega, but they are there are is no separation of this flow whenever it will exhibit with a velocity omega.

Now in this case the fluid on the cylinder surface that will moves at a circular velocity U should be is equal to r into r 0 into omega whereas, r 0 will be the radius of the cylinder and in that case the fluid sticking to the cylinder surface that will o of course, the viscosity of the fluid. Now the flow velocity at a given point on the cylinder you will see at an angle theta at point p as shown here in the figure that there will be a tangential velocity that will be denoted by v theta as well as the velocity that will be caused by this what is that angular momentum.

So, there will be a two components of this flow in that direction here in the tangential direction of this surface at point p that will be summation of v theta and U. Now what should be the v theta? We have already told that v theta will be is equal to 2U sin theta and then tangential velocity at point p it that will be vp that will be is equal to v theta plus u and that will be is equal to 2U sin theta plus r 0 omega.

So, this will be your tangential velocity component at point p whenever flow will be a flowing through a cylinder and at the surface of the cylinder at point here there will be a angular velocity omega and with this omega and what should be the that is tangential velocity. So, it will be as v theta and what is that u? So, by equation two you can get this tangential velocity here.

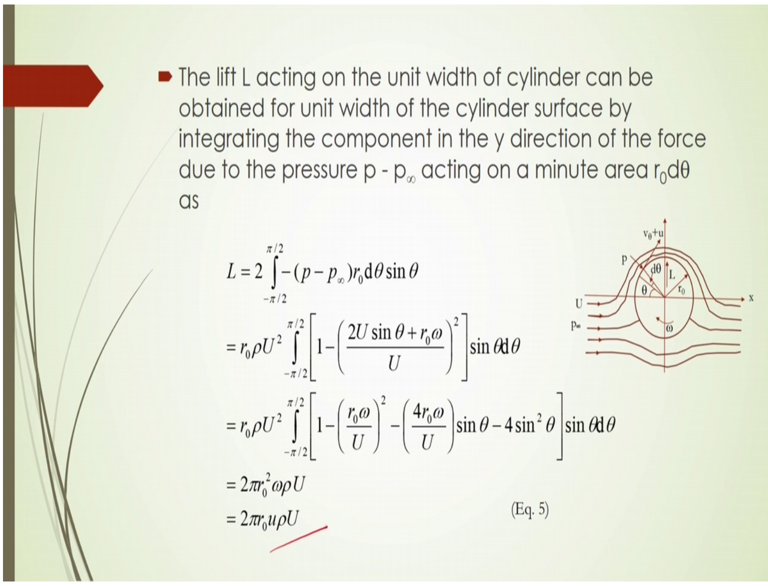
(Refer Slide Time: 11:45)



Now if we considered the pressure of the uniform flow as p infinity then the pressure at a given point on the cylinder surface it will be p. Now what should be the p value, then you can of course, have from this Bernoulli's equation here from this two points here p as well as at this point here. So, what should be the pressure at this point and what will be the pressure at this point? If the pressure at p is p then we can apply this Bernoulli's equation as here, then it will be p infinity plus rho U into U square that will be is equal to at this point this will be p plus rho y 2 into, here u square it will be the summation of that tangential velocity 2U sin theta plus r 0 omega that would be angular velocity that square.

So, after simplification we can have this p minus p infinity by rho U square by 2 that will be is equal to 1 minus 2U sin theta plus r 0 omega by U whole square. So, this equation four will give you the pressure distribution over the surface whenever fluid will be flowing over the surface of the cylinder. So, this equation of course, important for calculating the pressure how pressure will be distributed and based on that how energy will be distributed over the surface there?

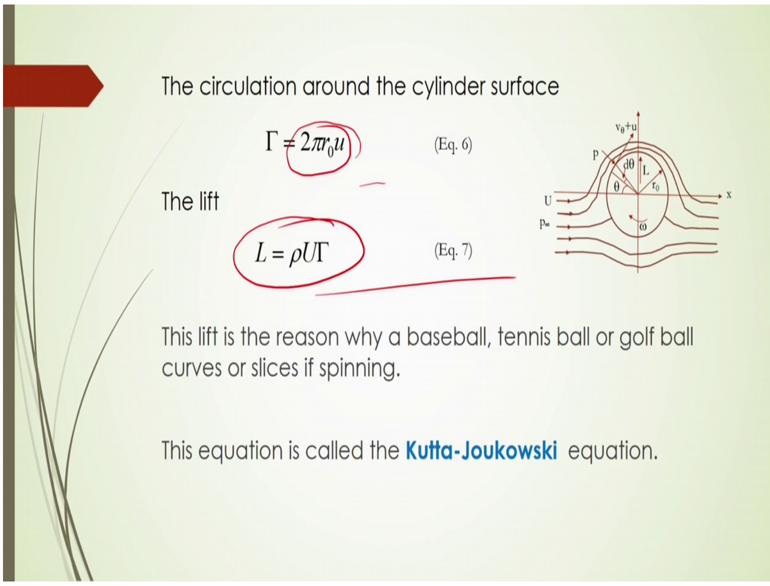
(Refer Slide Time: 13:15)



And again if we considered then if we have this pressure distribution or pressure there, then you will be able to calculate what should be the lift force there. So, in that case the lift L acting on the unit width of the cylinder if we consider can be obtained from the unit width of the cylinder surface by integrating the component in the y direction of the force due to the force that p minus p infinity acting on the minute area and it would be r 0 into d theta, as here L should be is equal to 2 that is the integration that is minus p by 2 plus pi by 2 that will be minus of p minus p infinity r 0 d theta into sin theta.

So, this will be your that is the lift components for the pressure force and from this pressure force if you have this component in the normal direction and after integration, then we can get the whole lift force for the whole surface of the cylinder and after simplification and integration here we are getting this finally, L should be is equal to 2 pi r 0 u into row U. So, equation number 5 will give you the lift force that would to be contributed by the pressure and the cylinder.

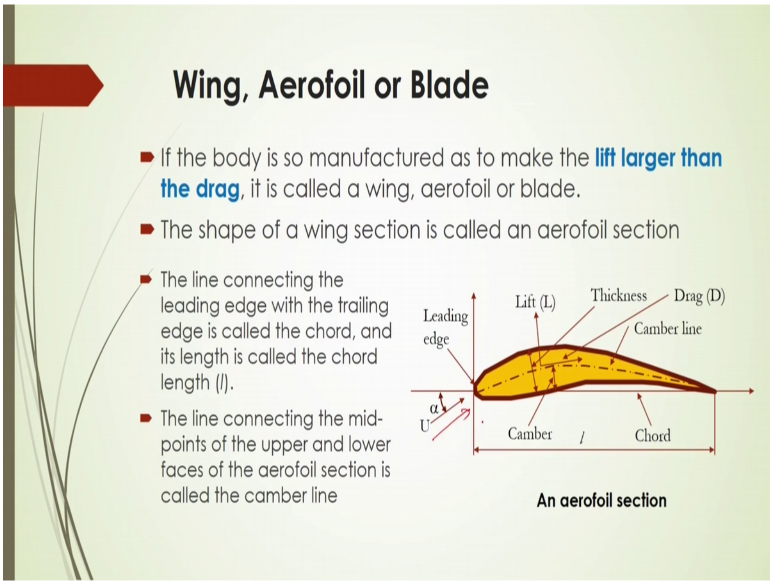
(Refer Slide Time: 14:36)



Now what should be the circulation around the cylinder surface that will be calculated if you know the circulation velocity that will be r 0 into u in that case here 2 pi r 0 into u that will be your circulation around. So, in this case this is simply by r 0 omega that circulation finally, it will be 2 pi r 0 u and then lift will be is equal to rho U into a circulation.

This rho U this is the definition for this lift forces for circulation, here this lift the here in this case it will be reason now for the base ball, tennis ball or a golf ball curves or slices if it is spinning there. Now this type of this equation number 7 it is called that Kutta Joukowski equation have from which you can calculate what will be the lift force if you know the circulation of the flow around the cylinder surface and if you know the free stream velocities there.

(Refer Slide Time: 15:41)

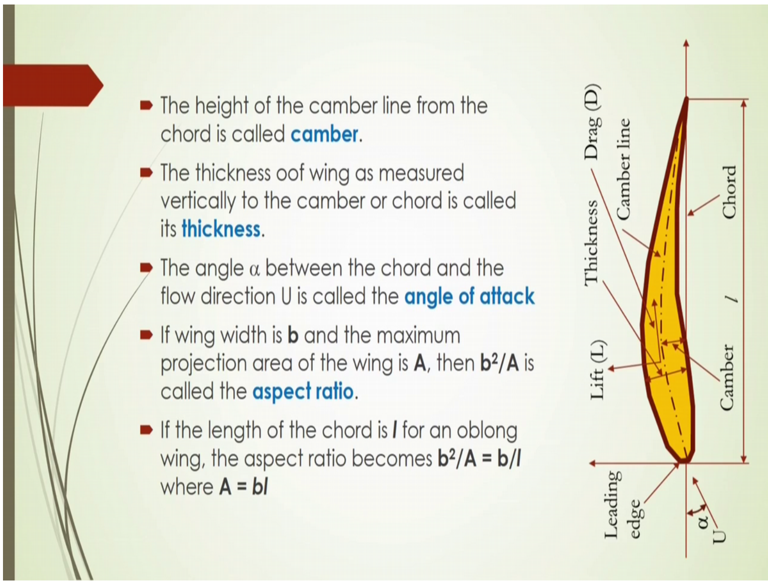


Now, if we consider the wing or aerofoil or blade, so in that case if the body is so manufactured as to make the lift that will be larger than the drag then it is called or wing or aerofoil or blade. That the shape of a wing section is called an aerofoil section, here the figure is shown in the slides the shape of wings that section is called the aerofoil section and the line that connecting the leading is with the trailing is called the chord and it's length is called the chord length. Here this to this length it's called that is this to this from this here it is called the line connecting the leading edge, with the trailing edge it will be called as chord length.

Now, the line connecting the midpoints of the upper and lower phases of the aerofoil section will be called as this will be called as camber. so this camber is nothing, but this what is the central line and what will be the is line here what will be the distance between these two edge line and centre line to be called as the camber.

Now other part of this that is aerofoil section is there here it will get if you consider a point here then in which direction this lift will be there and in which direction the lift that is drag force will be acting there and also what will be the thickness for that and camber line is there and if the velocity is acting on the surface of this aerofoil in this direction as shown here which makes to the central to the what is that chord line as alpha, then how the lift force will be calculated in that case there you have to consider.

(Refer Slide Time: 17:35)

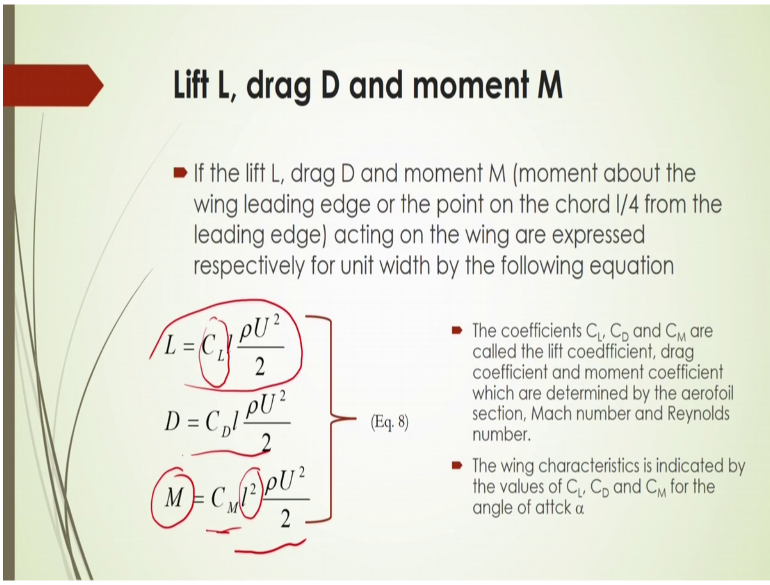


Now, in this case the height of the camber line from the chord is called camber and the thickness of wing as measured vertically to the camber or chord is called the thickness and here is shown in figure. The angle alpha between the chord and the flow direction U is called the angle of attack. If wing width is b and the maximum projection area of the wing is A, then b square by A which should be called as aspect ratio.

If the length of the chord is l for an oblong wing the aspect ratio becomes b square by A that will be is equal to b by l where A is equal to b into l. So, in this way we can calculate what should be the camber, what should be the thickness, what should be the angle of attack and what should be the aspect ratio? And this you have to calculate based on this definition.

And then we will be able to calculate what is the lift and as well as drag and moment M that will be acting on this aerofoil section. Now if the lift L, drag D and moment M, that is moment about the wing leading edges or the point on the chord l by 4 from the leading edge that will be acting on the wing, that expressed respectively for the unit width by the following equation here as given in equation number 8.

(Refer Slide Time: 19:03)



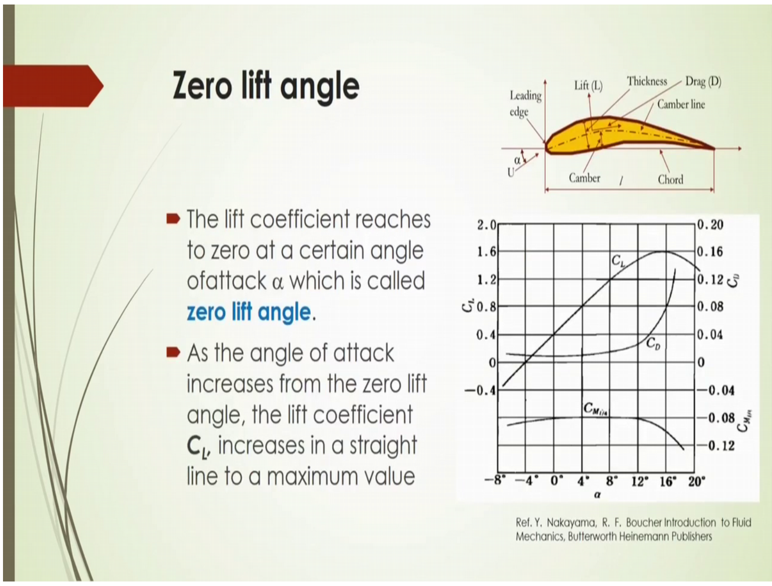
So, in that case L should be will be equals to as per definition earlier we have got this will be professional to the projectional area as well as what is that the kinetic energy of the fluid that would be flowing through this surface of the aerofoil section and then in that case you can have this L will be is equal to C L into l into rho U square divided by 2, here l is only for this area is calculated here unit width will be considered here.

So, in that case we can have the projectional area as l. So, the proportionality constant is called C L, the C L of course, it will be depending on the Reynolds number of the fluid flow and also drag force it will be as per that definition again C D into l into rho U square by 2. And what will be the momentum force that will be is equal to again it will be depending on the kinetic energy and also this what is the geometry that is cross sectional area here l square in this case, so C M it will be there is the proportionality constant.

So, C L, C D, and C M are the what is that lift coefficient drag, coefficient and the momentum coefficient you can say. The coefficient this C L, C D and C M are called the lift coefficient, drag coefficient and moment coefficient, of course, those will be determined by the aerofoil section and the mach number and the Reynolds number there this of course, the whether the flow will be above the sound velocity or not that will be very important if it is more than sound velocity.

Then what should be the mach number and based on which this drag coefficient and momentum coefficient will be considered and the wing characteristics is the indicated by the velocity of L, C D, C M for the angle of attack alpha in this case.

(Refer Slide Time: 21:05)



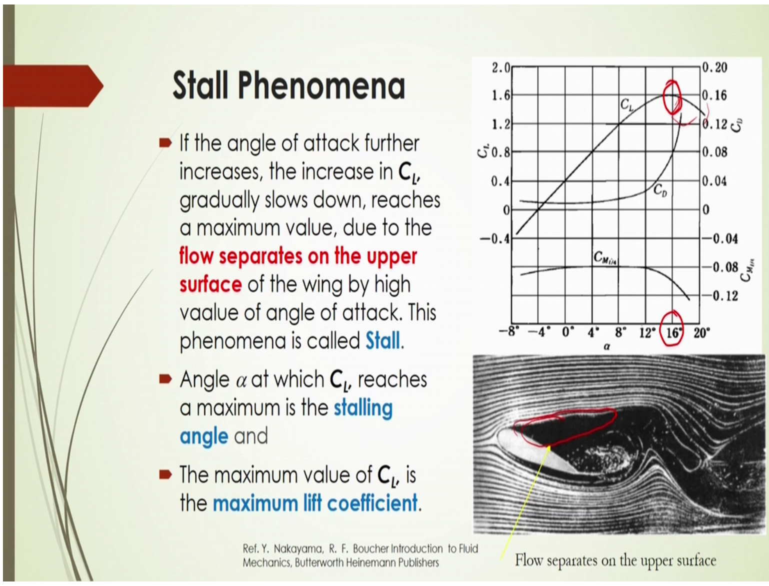
Another important section is that zero lift angle, what is that? Now the lift coefficient reaches to zero at a certain angle of attack, if you consider an alpha that is called zero lift angle and as the angle of attack increases from the zero lift angle the lift coefficient C L it will increase in a straight line to a maximum value as shown in figure here.

So, in this case you will see how the C L, C D and what is the C M? That is momentum coefficient are varying with respect to this angle here you will see if we increase the angle in that case you will see that the lift coefficient will be increased and at around 16 degree angle you will see there will be a maximum lift whereas, beyond that there will be change of this lift coefficient. Whereas you will see C D at that around 16 degree angle it will be increasing whereas, beyond this before this 16 angle; that means, if angle less than 16 degree, then in that case the C D will almost remain constant and also momentum coefficient also will remain almost constant.

But the momentum coefficient will decrease beyond this 12 degree and it will increase for drag coefficient beyond this 12 degree also. So, we can say that interesting point that at the zero lift angle means what at the 0 degree you will see the lift coefficient reaches to 0 and that will be represented as the zero lift angle. So, here in this case when this lift coefficient will be 0 that is minus 4 degree it is seen that if I have this minus 4 degree angle then you will see the lift coefficient will be 0.

So, that angle will be called as the lift that is zero lift angle there and also as the angle of the attack increases from the zero lift angle the lift coefficient increases in a straight line to a maximum value it will be up to 16 degree and beyond that it will decrease. Another important phenomenon of this that is aerofoil section you will see that is called stall phenomena, if the angle of attack further increases the increase in C L gradually slow down here in this case you will see there in the figure.

(Refer Slide Time: 23:55)



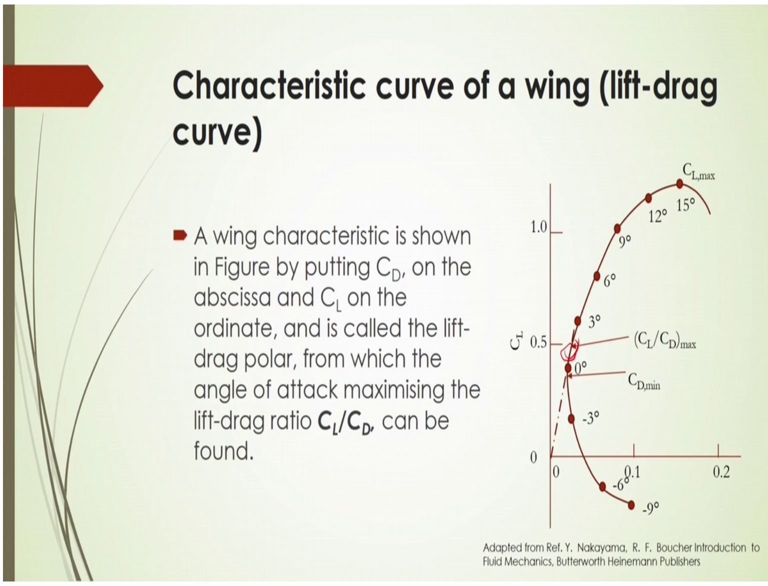
And it will reaches maximum value due to the flow separates on the upper surface of the wing by high value of angle of attack this phenomena is called stall. So, in this case you will see here interesting point that flow how this it will be separated from this wing and you will see and wing by high value of angle of attack in that case it will be more than 16 degree and the separation if this separation is happened this phenomena is called the stall. Now angle alpha at which the C L reaches a maximum is the stalling angle, it will be called as stalling angle that is maximum that is 16 degree here, it will be coming as the 16 that is C L here it will be maximum.

So, at 16 degree angle you will see this C L will be maximum, so this 16 degree angle it will be called stalling angle. At this angle you will see the flow will separates from the upper surface of the wing as shown in your figure and this maximum value of this lift coefficient is the maximum lift coefficient at this particular stalling angle. So, we can have this is the phenomena where we can say that flow will be separated from the upper surface of the wing of this aerofoil section and there will be a certain angle at which you can get this flow separation that phenomena is called stall.

Whereas the angle at which this stalling effect is the seen, it is called stalling angle and at this stalling angle you will get the maximum value of lift coefficient. So, we are getting this stalling angle as per this diagram here as shown in figure that 16 degree is your stalling angle whereas, the stalling effect will be happened when that is C L value is maximum of 1.6. So, we can have this stalling effect because of the separation of the flow from the wing.

Similarly another important point is that how characteristic curve of a wing that is lift drag curve can be obtained here? If we consider that wing characteristic in such a way that if we plot this C L curve by putting C D on the abscissa and C L on the ordinate, what will happen you will see that there will be angle of attack that will maximize the lift drag ratio of C L by C D. That here in that case C L by C D ratio will be maximum at point here 0.5 at this C L.

(Refer Slide Time: 26:51)

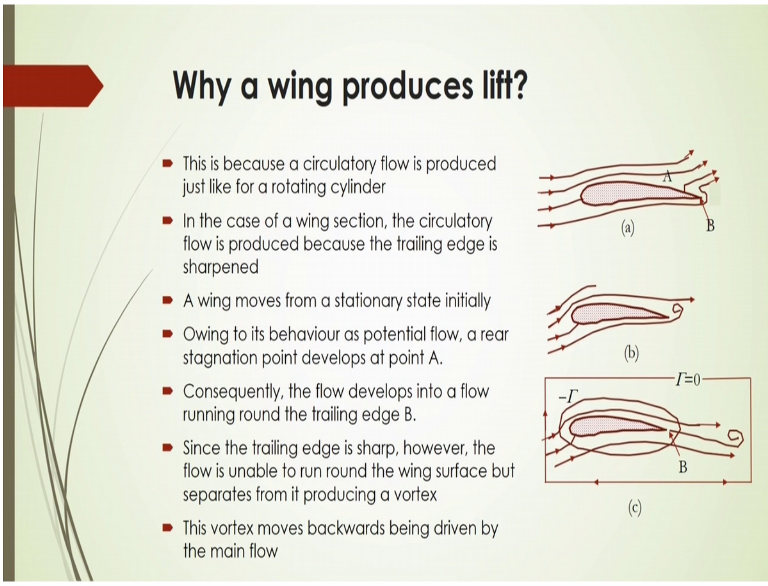


So, this will be when this maximum this lift drag ratio will be obtained that will be represented by this characteristic curve here. So, this characteristic curve you will see at a certain angle this you can have at a certain angle of this what is that is called angle of attack you will see this C L by C D if it is maximum it is coming that angle, that angle it will be represented as the characteristics curve angle or you can say lift drag angle.

So, that lift drag angle will be within between that 0 to what is that 3 degree in between there somewhere you will get because we are getting here in this curves the if we increase the angle there will be increase of C L and that will be maximum here whereas, the C D will be somewhere. So, when this C L by C D will be maximum that angle will be considered as the maximum attack angle for the maximum ratio of C L by C D.

And you will see there will be minimum value of C D at which you can get this from this curve also and this minimum phenomena of C D will get that you will see there will be a maximum [lid/lift] lift there.

(Refer Slide Time: 28:09)

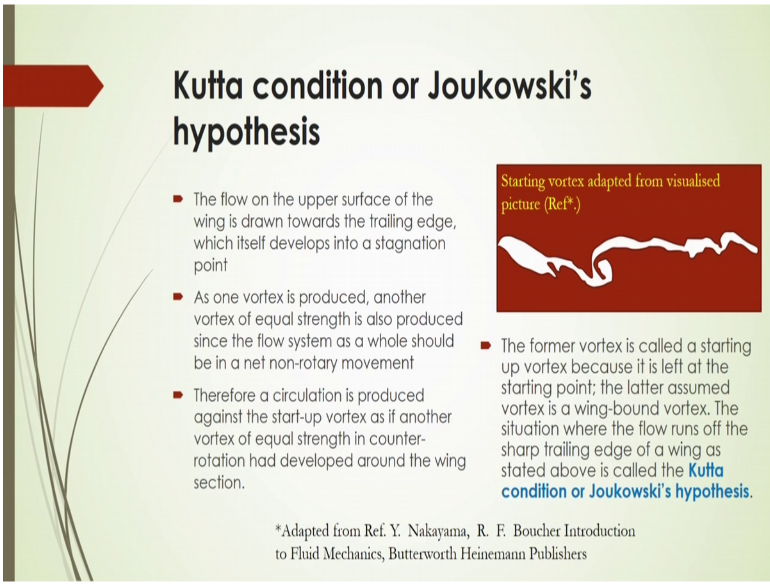


And also why a wing produces a lift then and it will be due to a circulatory flow that is produced just like for a rotating cylinder and in the case of a wing section the circulatory flow is that is produced because the trailing is here sharpened. Now, in that case the wing moves from a stationary state initially in that case owing to it's behaviour as potential flow and a rear stagnation point that will develops a point A and also you will see that because of that the flow develops into a flow running round the trailing is b.

And since the trailing is is sharps there at that point and the flow is unable to run round the wing surface, but separates from it producing a vortex there and this vortex moves backward that will be being driven by the main flow. So, here in this case you will see we are getting this three picture a b and c you will see very interesting that there will be a circulatory flow around this wing just like a rotating cylinder, that we got that ads and the circulation cell. And in this case you will see in this case of wing a section this circulatory flow actually produced because of the trailing is this trailing a sometimes is so sharpen that the smaller circulation cell will produce near about that is.

So, in that case that wing moves from a stationary state initially and owing to its behaviour as potential flow and you will see there will be formation of rear stagnation point at point A as shown in figure here and you will see since this trailing is sharp in that case sometimes; however, the flow is unable to run round the wing surface, but this flow will separates from its where the vertices are forms near about that wing surface and this forms sometimes will moves backward that will be driven by the main flow.

(Refer Slide Time: 30:39)



Now, another important that we got that some phenomena or that is by given by this Joukowski's equation there will be some hypothesis and that hypothesis will come based on the formation of vortex that is formed starting from the that is wing surface edge. So, in that case the flow on the upper surface of the wing actually is drawn towards the trailing is which itself develops into a stagnation point there. Now, as shown in figure you will see there will be the formation of a vortex that is started from the end of the edge of the aerofoil section or wing in that case, but, but a small circulation cell itself it will be forming from the stagnation point itself.

So, you will see that the flow on the upper surface of the wing whenever it will be drawn towards the trailing is, in that case the small circulation cell small circulation cell it will be broken down and it will be forms in such a way that after getting the reduced pressure at this edge there will be a formation of that is bigger that is what is that circulation cell and there you will see that (Refer Time: 32:05) will be coming to each other and formation of big (Refer Time: 32:09) there and that sometimes that will be collision and circulation cell will be enlarged there.

So, in that case the stagnation point sometimes it will forms that whenever the circulation cell will get struck somewhere in the wings, so that a stagnation point will be forming. Now when flow is a start immediately at the edge that is beginning edge there will be a formation of stagnant point and if suppose there is a that is the from the edge points that is the what is the sharpened end of this wing there sometimes you will see there be a some stagnation point there formation and over which there will be a flow that will moving beyond that because of the circulation and since the circulations are opposite in directions. So, they will form some stagnation zone over the point here.

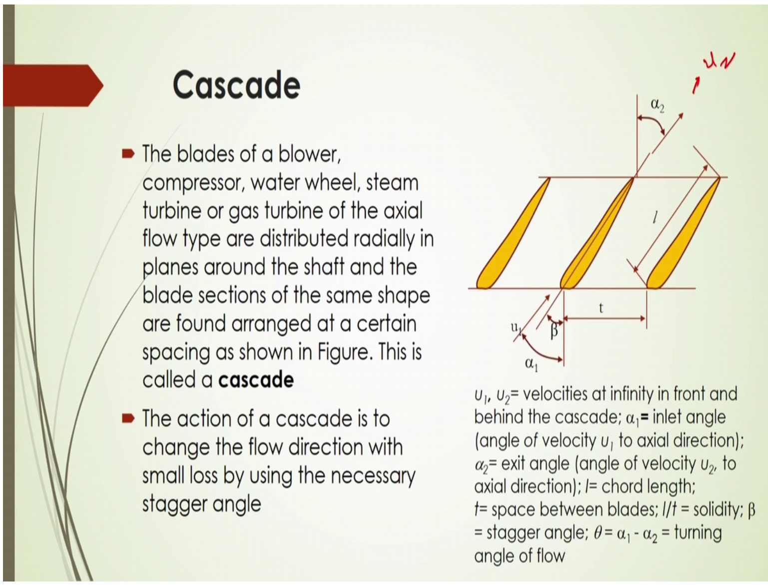
So, in this case you will see here this direction in this direction where this direction in this. So, whatever in this direction there will be a some stagnation region here also there will be a some stagnation region there. So, in that case the stagnation point will form because of that how it will be forming the vortex and the angle angular velocity that is phenomena that important point is that if the angular velocity is such that two angular that is circulation cell are coming to each other and they are forming the different direction of that angular portion then there will be a formation of some stagnation zone.

Now, as one vortex is produced the another vortex of the equal strength may be produced and because of that the flow stream as a whole should be in a net non rotary movement that is why. So, that is why you can say that there are two circulation cells two what is that vortex cells will be forming the stagnation zone in such a way that they will oppositely circulate to each other and that region that is the what is the gap between that the circulation cell there will be a formation of stagnation zone sometimes.

So, therefore, a circulation is produced against the start up vortex as if another vertex of the equal strengthen counter rotation had developed so around the wing section. So, that is why this circulate this stagnation point will be forming just by rotation of the vertex in opposite direction. The former vortex which will be called as starting up vortex because it is left at the starting point and the later assumed vortex is a wing bound vortex.

And in that case the situation where the flow runs of the sharp trailing edge of a wing as stated above it is called the Kutta condition or Joukowski's hypothesis. So, this phenomena it is actually based on this vertex phenomena and it is represented or it is first actually observed by this Kutta and it is called the Kutta condition or sometimes Joukowski's hypothesis it is called.

(Refer Slide Time: 35:21)



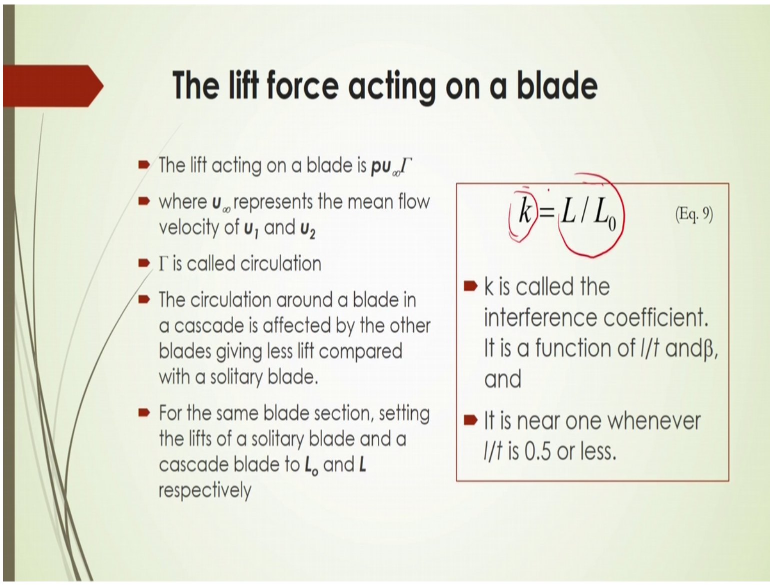
Now another important that what is this cascade; cascade is another important point how lift is the actually affected by this cascade phenomena here in this case. The blades of a blower the compressor, water wheel, steam turbine or gas turbine of the axial flow type are distributed radially in plane around the shaft and the blade sections of the same shape are found arranged at a certain spacing as shown in figure this is called cascade and the action of a cascade is to change the flow direction with small loss by using the necessary stagger angle.

Here in this case you will see U 1 and U 2 it is shown that in this figure that here the velocities at infinity in front and behind the cascade and also you will see here this is U 2 here U 2 this is U 1. So, these are the velocities at infinity in front and behind the cascade and alpha 1 is equals to inlet angle of velocity U 1 to axial direction in this case and alpha 2 here as shown in figure that it will be exit angle that is the angle of velocity U 2 to axial direction and l that will be a chord length and also you will see that t there will be a certain space of this cascade.

So, it will be space between blades for cascades you will see and l by t that is called solidity and beta angle is there whereas, this axis will make with a vertical direction it will be called as stagger angle and theta is equal to alpha 1 minus alpha 2 it is denoted by turning angle of flow. so, we are having this cascade phenomena how this different angles of this flow velocity will be with this axis of the cascade there.

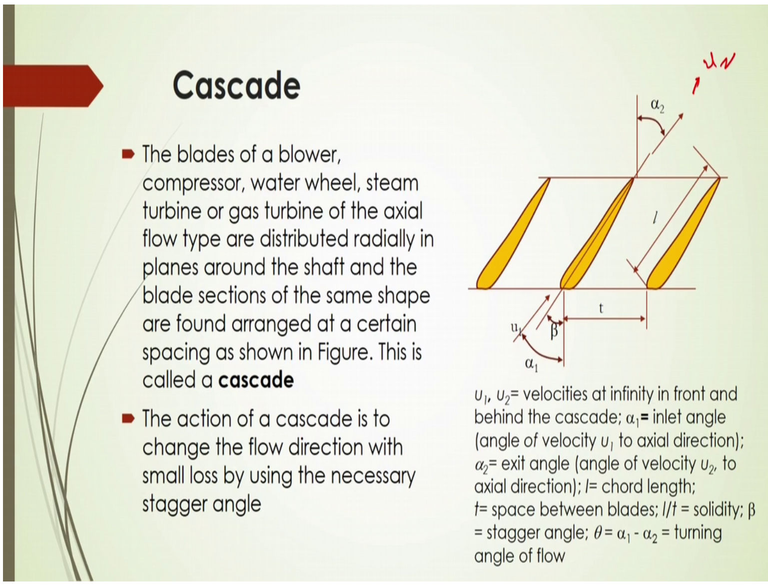
Now if you consider the lift force acting on this cascades how it will be there the lift acting on a blade is that is rho U infinity into a gamma where rho is the density of the fluid and U infinity is the uniform velocity free stream velocity and gamma is called circulation and in this case the circulation around a blade in a cascade that will be affected by the other blades giving less lift compared to a solitary blade and also for the same blades you will see set that is the setting the lifts of a solitary blade and a cascade blade to L 0 and L respectively in that case we can have this interference coefficient denoted by k that will be is equal to L by L 0.

(Refer Slide Time: 38:22)



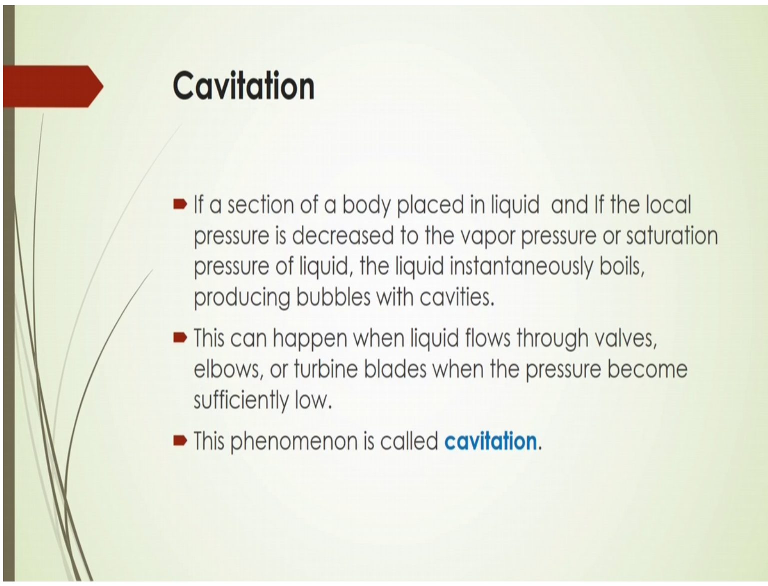
So, this will be a function of l by t of course, this what is that chord length to the thickness of this blade there are spacing of this blade and also staggering angle. So, in that case it is near one whenever l by t will be is equal to 0.5 or less.

(Refer Slide Time: 38:47)



So, this interference coefficient is important to design this blade angle blade and also for the cascade shape and then what should be the positions for that blade based on this thickness and as well as that chord length and based on this velocity how it will be calculated and it will be designed here.

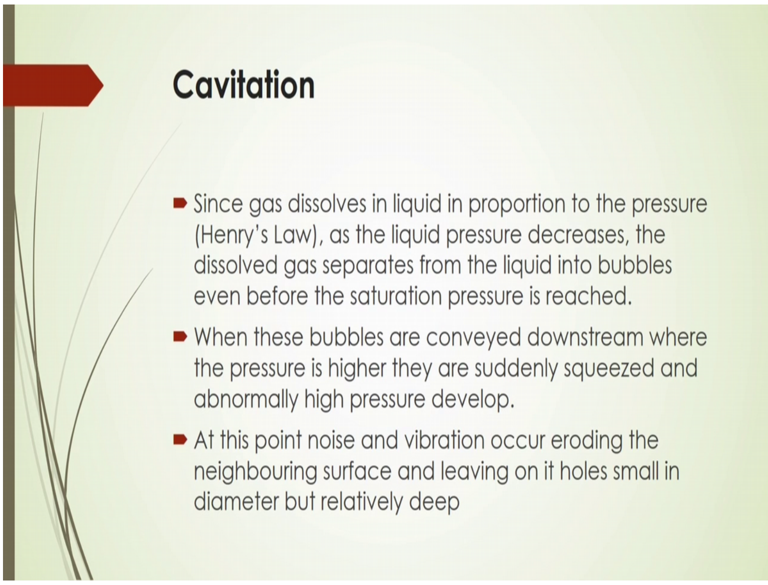
(Refer Slide Time: 39:08)



Now, next part of this what is the cavitation, now cavitation you will see it is important because sometimes whenever fluid will be flowing over this solid object you will see there will be a sudden change of pressure over the surface. So, in that case if a section of the body placed in liquid and you will see that if the local pressure is decreased to it's vapour pressure or saturation pressure of the liquid you can say that the liquid instantaneously will boils and in that case there will be a formation of bubbles that will be called cavity and this can happen when the liquid flows through the valves and elbows or turbine blades and also in that case the when the pressure become sufficiently low and this phenomenon is called the cavitation.

So, basically the cavitation happened when you will see if in any section the the body that will be placed in the liquid if the local pressure is reduced to the vapour pressure of the liquid, then you will see there will be a formation of bubbles and that bubbles will sometimes hamper the flow and also the valve and elbows and turbines whenever fluid will be flowing through this section or through this small bodies. So, in that case suddenly the pressure will sufficiently will become low and they are inside that valve or elbows there will be a formation of cavity or that is bubble and that will obstruct the flow of phenomena and also the what is that there will be a different what is that loss of that energy and also you will see there will be a corrosion erosion and also that there will be breakage of the or that is malfunctioning of these valves or other equipments there.

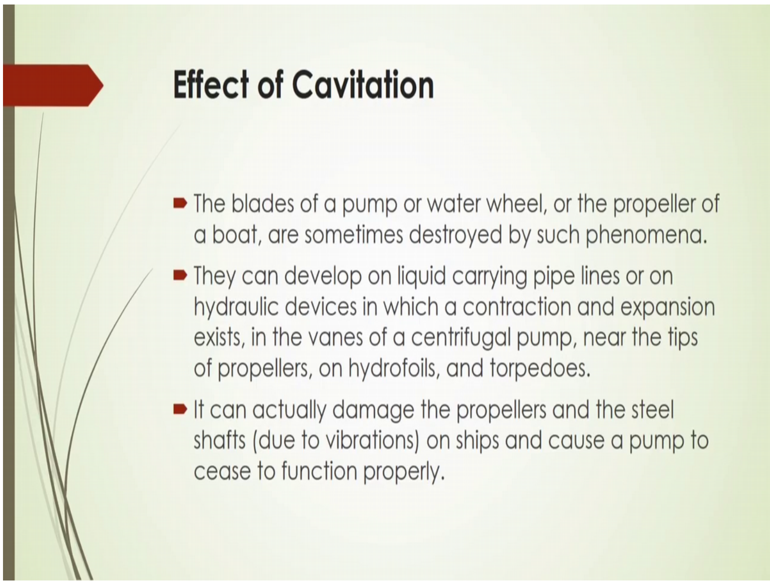
(Refer Slide Time: 41:13)



Now, in that case if suppose the gas dissolves in the liquid in proportion to the pressure as per Henrys law, as the liquid pressure decreases, so the dissolved gas separates from the liquid into bubbles and even before the saturation pressure is reached there. Now if this bubbles are that is conveyed downstream where the pressure is higher they are suddenly squeezed and abnormally high pressure will be developed there.

Now at that point there will be a formation of noise, vibration will be there and also erosion will be there neighbouring the surface and also you will see there will be formation of small holes in the diameter and it will be sometimes it will be deep also because of this variation of this pressure and formation of the bubble there.

(Refer Slide Time: 42:02)

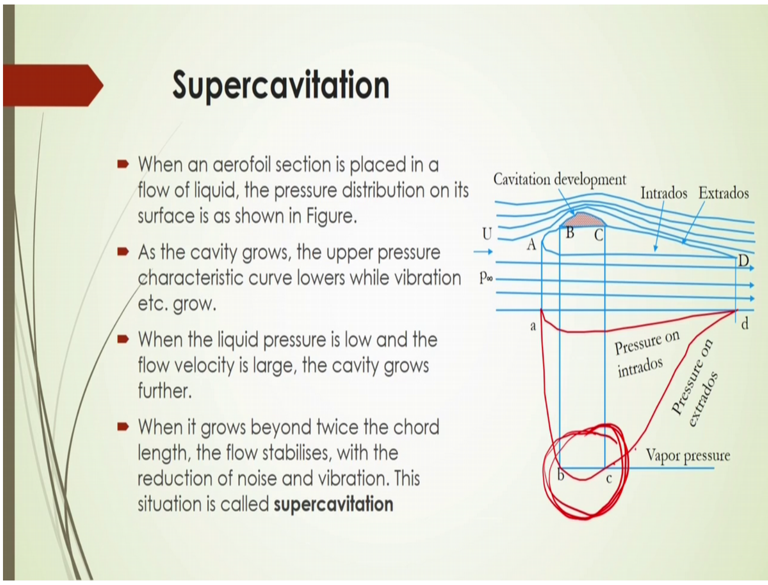


And you will see sometimes this blades of the pump or water wheel or the propeller of the boat sometimes destroyed by such phenomena. So, they can develop a on liquid that will be carrying through the pipe lines or on the hydraulic devices in which a contraction or sometimes expansion will be there. So, because of which that in the vanes of a centrifugal pump near the tips of the propellers, there will be a formation of bubbles and in that case there will be a formation of hydrofoils and the torpedoes there.

And it can actually damage the propellers and the st[eel]- steel shafts due to the vibrations on ships and cause a pump to cease to the function properly. So, that is why I told that if the cavitation is formed it will sometimes malfunctioned in it's work condition. So, in that case you have to consider this effect of this vapour pressure at which condition this vapour pressure will be forming and you have to know the pressure distribution whenever fluid will be flowing through that you have to see whether this pressure is suddenly becoming low or not.

Whenever it will be becoming low as this boiling point pressure then there will be a problem of this flowing of flow or through this of pipe sections and even also other hydraulic devices system. So, in that case we have to design the devices in such a way that when that the range of the pressure you have to consider and also the when this pressure will forms that cavitation and whether that cavitation can be avoid or not by just changing the design phenomena, design criteria based on this pressure development.

(Refer Slide Time: 43:51)

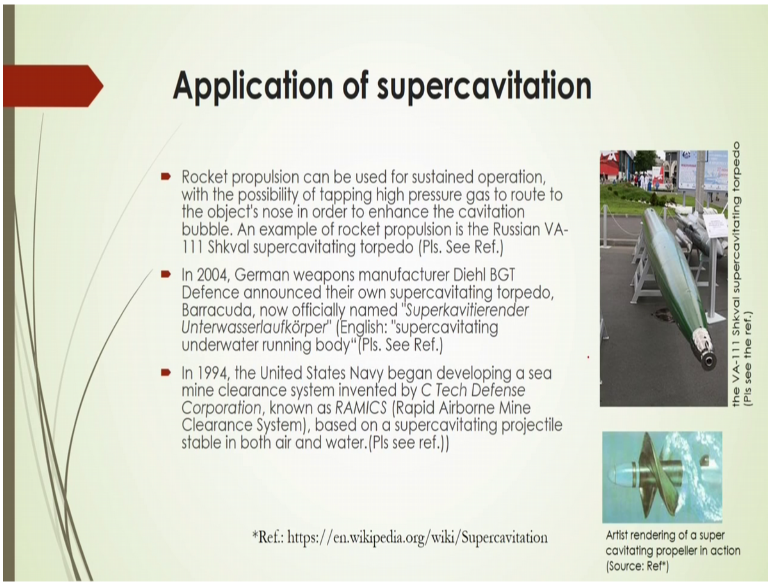


Now, what is that supercavitation it is also important, now if you are considering an aerofoil, so when an aerofoil section is placed in a flow of liquid in that case you will see there will be a pressure distribution on it surface as shown in figure here, there will be a pressure distribution as shown here this is by red line it is that the pressure distribution over there. Now, as the cavity grows if suppose there is a formation of cavity for this lowering pressure here what will happen, the cavity whenever it will be growing up then the upper pressure characteristics curve lowers the vibration etcetera.

So, in that case the cavity grows the upper pressure characteristics curve in that case it will lower and also vibration also will grow there. So, when the liquid pressure is low and the flow velocity is large the cavity grows further. So, when it grows beyond the twice the chord length you will see important that if this cavity; if this cavity is larger than the chord length, that is if it is twice the chord length the flows will stabilize and with the reduction of noise and vibration and this situation will be called as supercavitation.

So, you have to form this cavitation if is there any cavitation is formed then you have to make it twice of the chord length. So, that the flow will be stabilized and there will be a reduction of the noise and also the reduction of the vibration and so this is the phenomenon based on which you can design this aerofoil where then you can get this minimum or you can say almost negligible amount of noise and vibration there. So, you have to calculate the super critical condition and this super critical pressure or super that is cavitation pressure based on which what there will be no formation of cavity there.

(Refer Slide Time: 45:51)

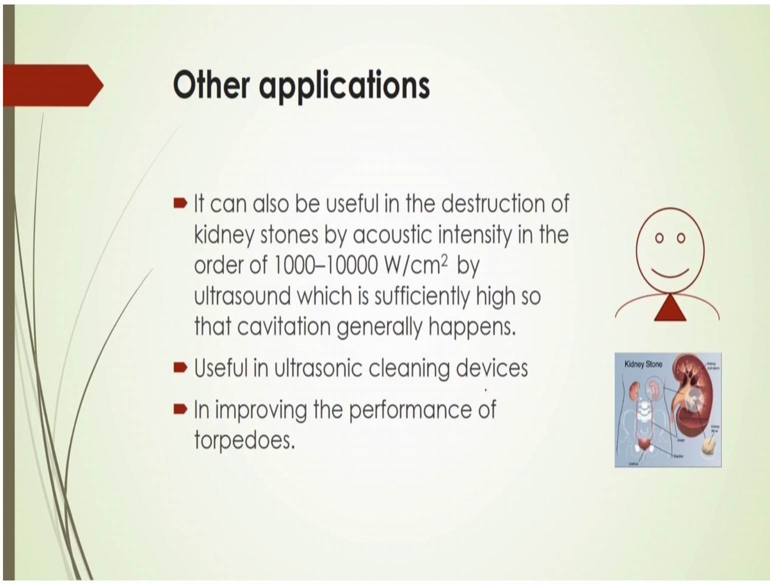


Now, where then this supercavitation can be applied you will see whenever you are going to design a rocket propulsion, so in that case this supercavitation phenomena is considered. So, so rocket propulsion can be used for sustained operation with the possibility of tapping high pressure gas to route the objects noise in order to enhance the cavitation bubble.

An example of the rocket propulsion in Russia that they have developed that it will be VA-111 that is Shkval supercavitating torpedo here shown in figure and also you will see that in 2004 German weapons manufacturer that is the Diehl BGT defence announced their own supercavitating torpedo and they have designed this torpedo based on this supercavitation condition.

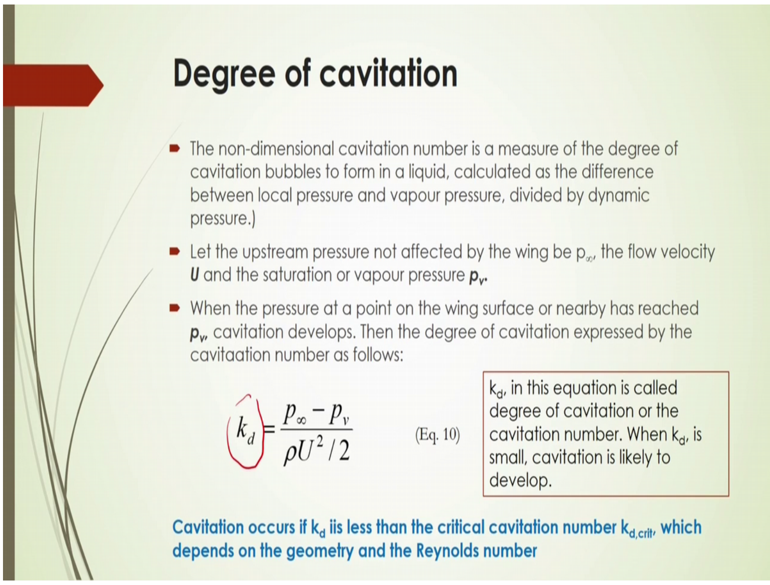
Also in United States 1994 Navy began developing a sea mine that is clearance systems invented by C Tech Defence Corporation it is known as RAMICS that is Rapid Airborne Mine Clearance System based on the supercavitating projectiles stable in both air and water. So, supercavitation actually applied for this designing this the rocket even other even manufacture in the defence section.

(Refer Slide Time: 47:27)



Now other applications it can also be useful in the destruction of kidney stones by acoustic intensity in order of what is that 1000 to 10000 watt per centimetre square by ultrasound which is sufficiently high. So, that cavitation generally happens. So, in this case you have to produce some small bubbles by this acoustic intensity and that will be used for taking up that kidney stones. And also it is useful for ultrasonic cleaning devices there and also it is used for improving the performance of the torpedoes.

(Refer Slide Time: 48:06)



Now, how to calculate this degree of cavitation, when that cavitation will be occurred and what will be the condition for that when I we should not actually propose or we should not suggest to design beyond that condition for the equipment there? So, the non dimensional cavitation number is sometimes is used to actually measure the that is degree of cavitation and to form in a liquid. Now that will be calculated as the difference between the local pressure and the vapour pressure that will be divided by the dynamic pressure.

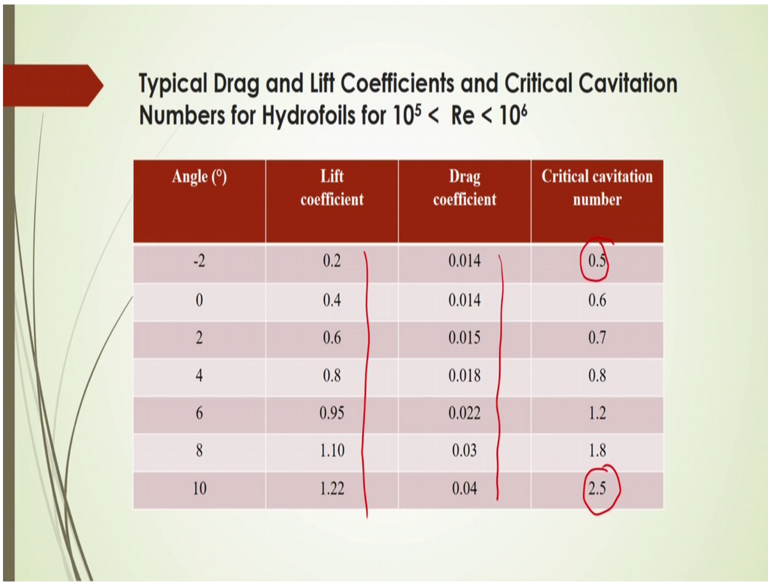
Now let the upstream pressure not affected by the wing be here as p infinity and the flow velocity is U and the saturation or vapour pressure of the liquid is considered as p v. So, in that case when this pressure at a point on the wing surface or you can say nearby has reached this vapour pressure p v in that case cavitation will develop and then the degree of cavitation that is how much that is what is the intensity of that cavitation whether this pressure difference will be a less than or greater than that kinetic energy supplied there.

And based on which the degree of cavitation will be actually calculated and it is generally expressed by this cavitation number and this cavitation number is denoted by this k d that will be defined as p infinity minus p v by rho U square by 2. So, this k d in this equation is called the degree of cavitation or cavitation number you can say when k d is small then cavitation is likely to develop.

So, you have to consider that k d in such a way that it should be not less than what is that if it is a less than the critical cavitation number which depends on the geometry and the Reynolds number. So, you have to consider this cavitation number in such a way that it should be small enough if it is there, but the kinetic energy also should be considered, but you have to have the optimum value, so that p infinity should be always p v.

So, if infinity is greater p v; that means, here you will see that p infinity will be kinetic what is the kinetic energy you have to multiply this k d and then you have to add the vapour pressure then only you have to this pressure. So, always that pressure p infinity will be is equal to p v plus k d into rho U square by 2 if you are maintaining then you can get the flow without cavitation.

(Refer Slide Time: 50:54)

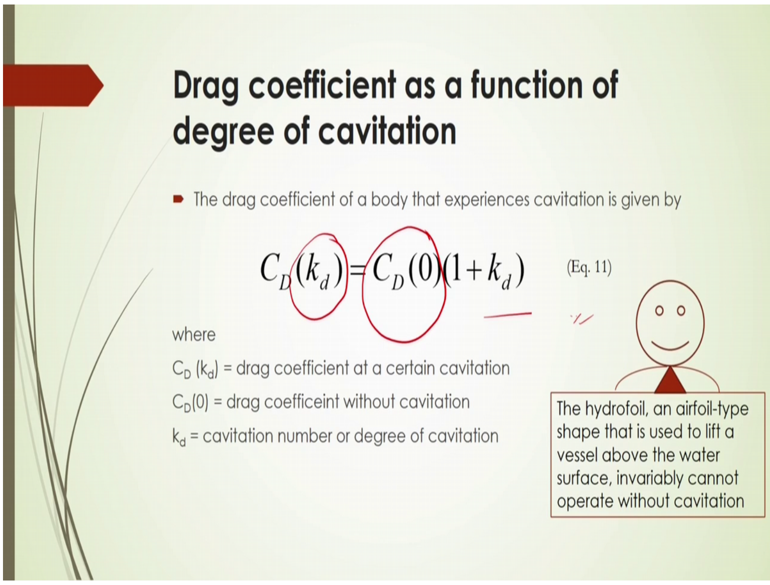


Angle (°)	Lift coefficient	Drag coefficient	Critical cavitation number
-2	0.2	0.014	0.5
0	0.4	0.014	0.6
2	0.6	0.015	0.7
4	0.8	0.018	0.8
6	0.95	0.022	1.2
8	1.10	0.03	1.8
10	1.22	0.04	2.5

Now, some typical drag and lift coefficients and critical cavitation number for hydrofoils within the range of Reynolds number here it is given. So, in this case if the angle of attack is there if it is minus 2, 0, 2, 4, 6, 8, 10 accordingly the lift coefficients are given here 0.2, 0.4, 0.6 up to 1.22 here for at up to angle is 10 and in that case respective drag coefficients are also given this and what will be the cavitation number in this case 0.5 and also up to 2.5 there for this angle of cavitation. So, it is important that if your angle of attack is 0 in that case you can have this critical cavitation number as 0.6. So, there will be a certain value of this cavitation number and because of which then you can have the cavitation.

So, if there is no cavitation what does it mean that k should be equals to 0 and in that case you will see that p should be is equal to p v. So, at that critical condition that you will see that there will be what is that p almost will be equals to that is vapour pressure, so in that case k d will be is equal to 0.

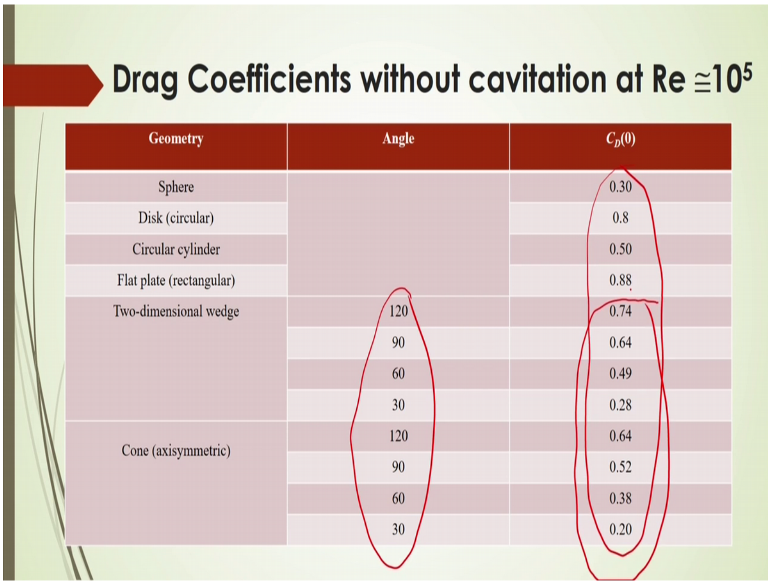
(Refer Slide Time: 52:19)



Now, if we represent this drag coefficient as a function of degree of cavitation how it can be there? The drag coefficient of a body that experiences the cavitation that will be given by this equation number 11 that will be C D into k d C D that is as a function of k d naught into C D, that will be is equal to into C D C CD without cavitation into 1 plus k d.

So, interesting that one is C D that drag coefficient at a certain cavitation and C D is 0 that will be drag coefficient without cavitation and k d is equal to this cavitation number or degree of cavitation here. So, in this case you have to remember that the hydrofoil and airfoil types shape that is used to lift a vessel above the water surface invariably cannot operate without cavitation there.

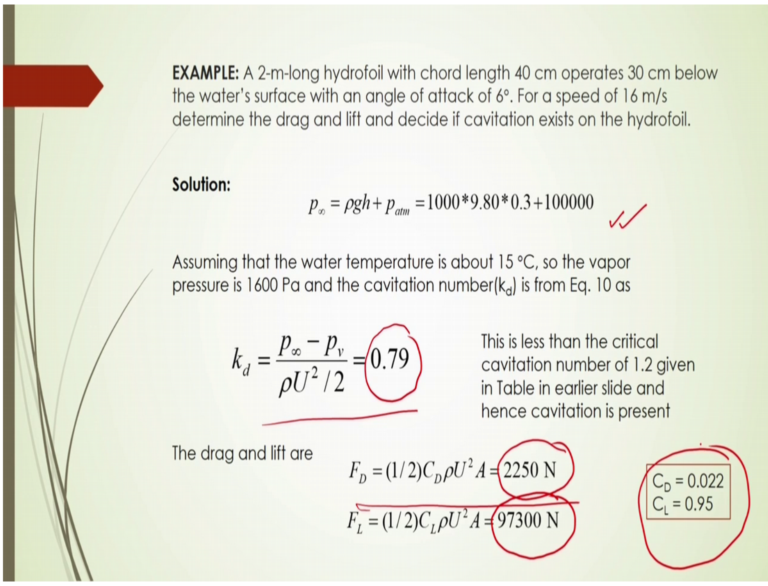
(Refer Slide Time: 53:16)


Drag Coefficients without cavitation at Re ≅10^5

Geometry	Angle	$C_D(0)$
Sphere		0.30
Disk (circular)		0.8
Circular cylinder		0.50
Flat plate (rectangular)		0.88
Two-dimensional wedge	120	0.74
	90	0.64
	60	0.49
	30	0.28
Cone (axisymmetric)	120	0.64
	90	0.52
	60	0.38
	30	0.20

Now, drag coefficient without cavitation at Reynolds number of 10 to the power 5 here some that is drag coefficient without cavitation that can be obtained for this geometry; that means, like a sphere here disk and circular cylinder, flat angle, two dimensional wedge and cone for different angle of attack in that case respective value of drag coefficient without what is that cavitation it is there.

(Refer Slide Time: 53:48)



Now, with cavitation then how it will be there, in that case how to calculate? Let us do an example in this case [luppose/suppose] suppose a 2 meter long hydrofoil with chord length of 40 centimetre that will operate 30 centimetre below the water surface with an angle of attack of 6 degree. Now, in that case for a speed of 16 meter per second in that case determine the drag and lift and decide cavitation exists on the hydrofoil or not.

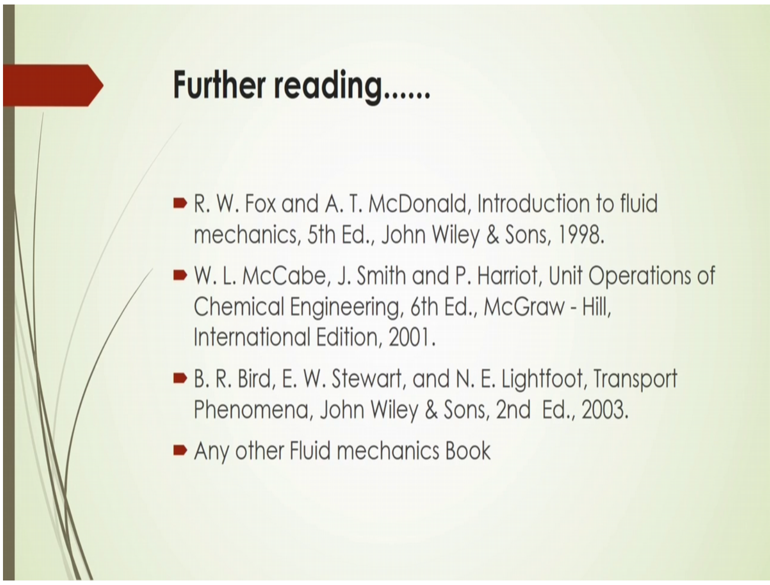
So, in that case this p infinity you have to first calculate that will be rho gh plus p atmosphere at this surface of this water what will be the pressure in this case this will be your pressure and then assuming that the water temperature is about 15 degree centigrade at that temperature vapour pressure will be 1600 Pascal and the cavitation number based on this equation number 10 we can have this k d will be is equal to p infinity minus p v by rho U square by 2 that will be is equal to 0.79.

After substitution of this p infinity and p v there we are getting this k d will be equals to 0.79. So, this is less than the critical cavitation number of o1.2 that is given in table here earlier this cavi[tation]- 1.2 this one. So, in case we can say that hence the cavitation will be of course, present during this flow and the drag and lift force accordingly it would be calculated based on this formula here 1 by 2 into C D into rho u square A. So, based on which we are getting this drag force is 2250 and F L will be is equal to 97300 Newton whereas, C D and C L are given as 0.022 and 0.95 respectively.

So, based on this example whether there will be cavitation will occurs or not very simple way you can calculate there. So, what you have to see that whether this critical cavitation number is there or not, now if this k d value is less than a critical cavitation number then you of course, will expect that there will be a cavitation. So, always this cavitation number should be greater than this critical cavitation number, so that you have to remember here.

So, critical cavitation numbers are given here at this condition there will be a formation of cavitation just start here in this case and then you have to of course, follow that this cavitation number whatever cavitation number as per your design it will be greater than this critical cavitation number.

(Refer Slide Time: 56:38)



So, you can further read this books for further understanding of this phenomena of this lift and cavitation; so.

Thank you for this lecture today.