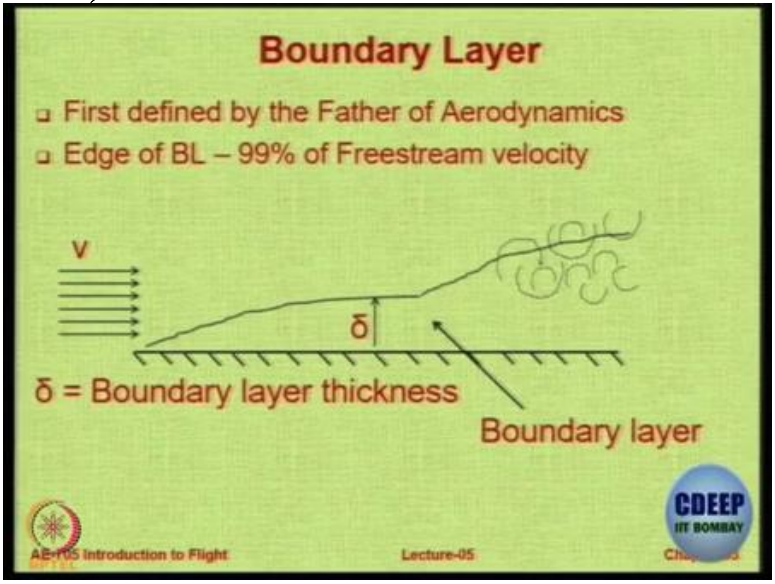
Introduction to Flight
Professor Rajkumar S. Pant
Department of Aerospace Engineering
Indian Institute of Technology Bombay
Lecture 04.2 – Introduction to Boundary Layer

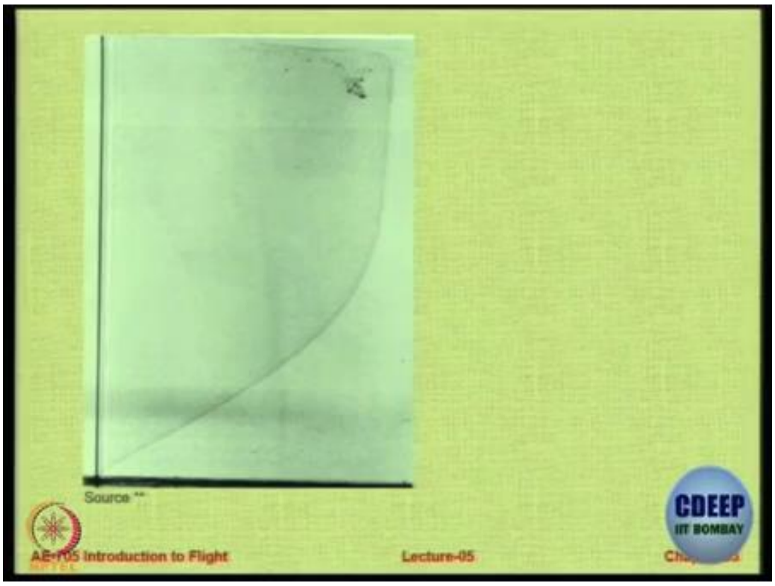
(Refer Slide Time: 0:32)



The region in which the viscous flows are predominant is boundary layer and the other region is the normal or the non-boundary layer. So, this is the contribution by the father of aeronautics Ludwig Prandtl and he also gave a definition, where does the boundary layer end? The boundary layer ends when the flow stream velocity inside the boundary layer is almost equal to the free stream velocity.

He gave a number of 99 percent because if you want to go to 100 percent, then you have to really do a lot of calculations. So, that is why he said 99 percent. So, here is a flat plate, here is a steady uniform flow of velocity 'V' and area gets built up which then also changes. So, after some time you can see that the boundary layer thickness increases very rapidly. So, this particular place, this particular thickness is called as the boundary layer and the thickness is called as the boundary layer thickness and notice the boundary layer thickness delta is not constant, it is continuously increasing up to some point and then there is a rapid increase beyond some point. So, the boundary layer thickness does not remain constant as the flow mark number changes.

(Refer Slide Time: 1:40)



This is a very interesting image, here we see a platinum wire on which again we are passing current. So, the water gets reacts with the electrical current and hydrogen is released and these hydrogen bubbles, they travel at the speed of the local flow. So notice that almost at the bottom the bubbles are at almost zero velocity. It is not exactly zero because the wire is touching the surface, so at that touching point there will be some variation. But, notice that as you go above the surface, so this line basically is the locus of the hydrogen bubbles which have been released from the wire and after some time 't' they are at that point. So, then what is this line? Pathline, timeline, streakline, streamline?

Yes, take the mic and justify your answer.

Student: It is pathline.

Professor: Why is it a pathline?

Student: Because, it shows the path of the particles moving.

Professor: No. No, this is not one particle moving along that line, if it was one particle moving along that line, I would say it is a pathline. This is a locus of literally thousands of bubbles which have started from time 't' equal to zero from this pipe, from this wire and after some time we have taken a picture. So, what is it? Take a mic.

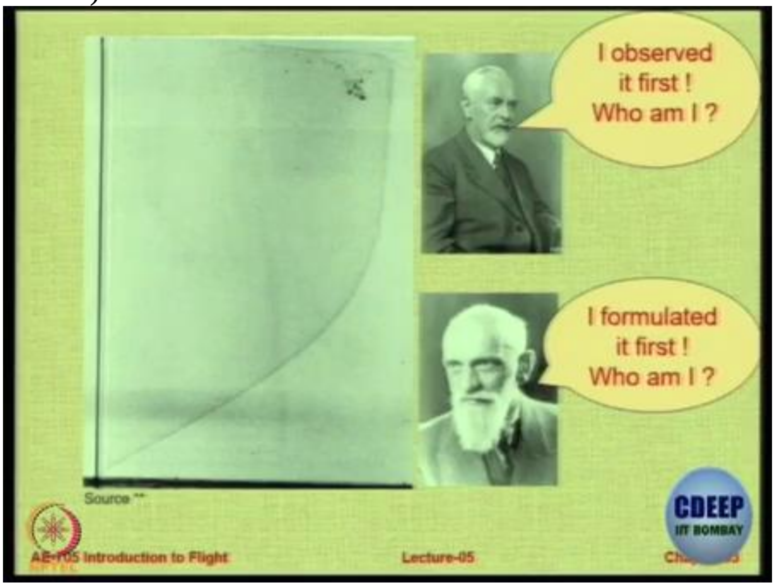
Student: It is a streakline.

Professor: Why is it a streakline?

Professor: That is not true. What is happening, I will repeat here. All along this particular wire, hydrogen bubbles, different bubbles, hundreds of them have been created, their diameter is very small, so therefore you cannot see the diameter and at time ‘t’ equal to zero we pass, current bubbles are created. At time t equal to 3 seconds, let us say we have taken this picture. In those 3 seconds the front that you see is the locus of the position of the bubbles all of which left the wire at time t equal to zero. So, what is it?

Students: Timeline.

(Refer Slide Time: 4:38)



Professor: You have to understand the four basic concepts, ok. So, first of all, who is the guy who observed it first? And he observed it but he could not formulate it, he could not do any calculations to do it. Ok. So, the gentleman on the bottom is the first person to formulate it to give us a theorem. So does anybody know? The one on the top of course is Ludwig Prandtl. I told you already about him but who is the person below who studied what Prandtl observed and did a formulation? Yeah, take a mic please. There is a theorem after his name, take a mic.

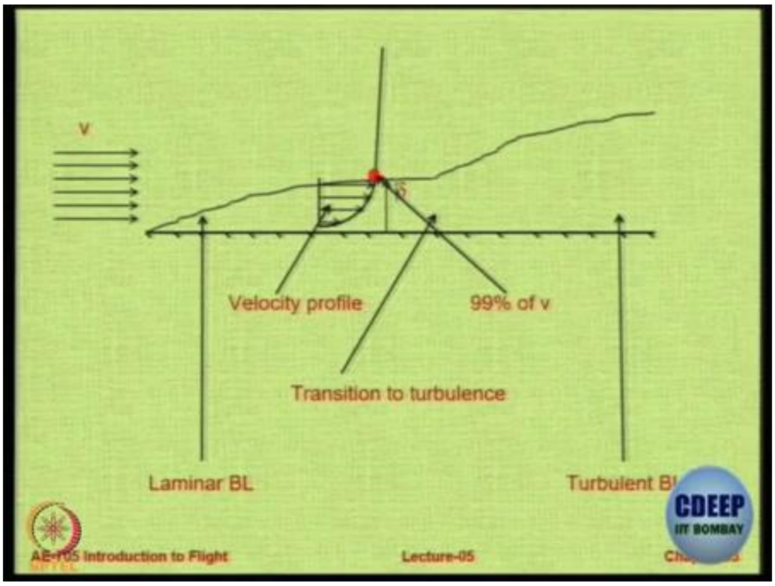
Student: He is Von Karman.

Professor: No, no, no he is not Von Karman; Von Karman came very much late. He is not Von Karman. Anybody else knows?

Student: I think so Blasius.

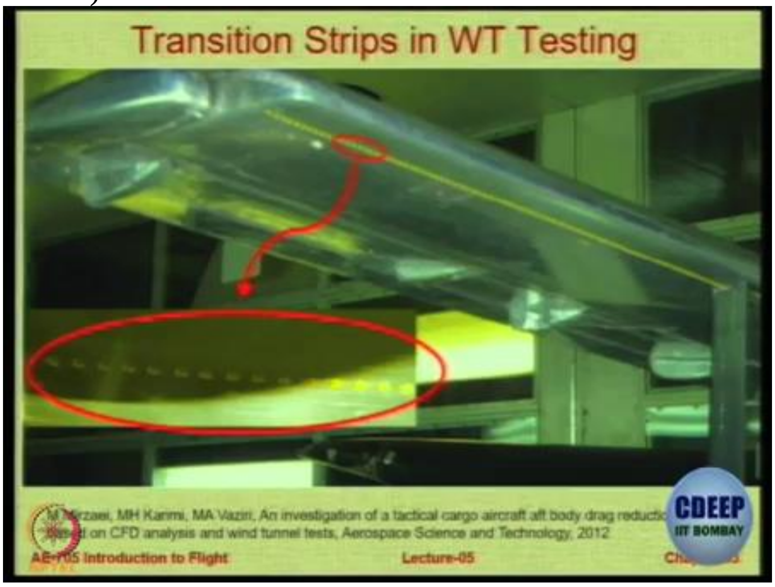
Professor: Blasius. We have a Blasius theorem about the boundary layers, ok.

(Refer Slide Time: 5:35)



So, here is a full picture of the same thing that we saw earlier, now I have just injected the profile. So, you have a laminar boundary layer in the beginning till there is a transition, and you have these profiles and that red dot basically is the place at which the velocity becomes 0.99 times v infinity, that is why it is called as Delta, the thickness.

(Refer Slide Time: 5:57)



Ok. Alright now. Now, I have some interesting things to share with you. This is a photograph of a wind tunnel model for a cargo aircraft. So, a cargo aircraft wing, there was an attempt to do some modifications on the rear side, aircraft aft body drag reduction based on CFD analysis etcetera. This is a wind tunnel model and what do we observe below the wind tunnel model is that at the bottom of the wind tunnel model, on the bottom surface you have these yellow dots, ok. And in the close up, you can see these are actually clearly visible as projections. So, my question is, in this wind tunnel test why do you think these yellow dots have been inserted?

Is it because the wing itself had these projections or is it because there is a need to put them? So, now if you have understood what I have discussed so far, you should be able to think about it. So, let me give you a hint.

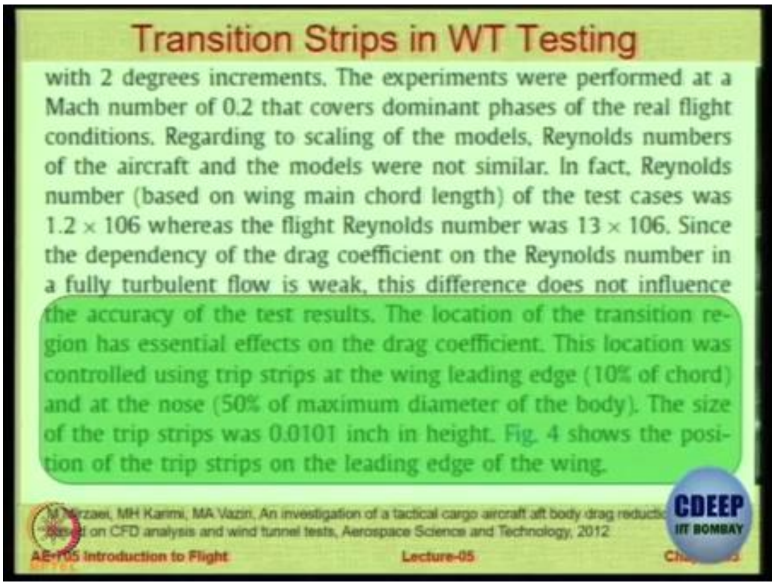
Number 1 – These yellow dots are not there in the actual aircraft. They have been inserted forcibly. And if you do not put these, then the results obtained, the aerodynamic results obtained for this particular wind tunnel test, they will never match with that of the actual aircraft. So, it is intentional. So question is- What are these and why they have been put? Yes.

Student: Sir, because of length scale in an actual aircraft it will be much longer, so the flow may be already turbulent by the time it reaches that point. But in wind tunnel model it will not be turbulent, so we have to chip it in.

Professor: Ok. So, that is the right answer. These have been intentionally put because in the large scale aircraft, because of the surface texture or because of the dimension, and because of the Reynolds number that is applicable in large scale flow. Remember, the full scale aircraft, it will have a different value of Rho, V, Mu by L.

Ok, because it will be at a higher speed. In the wind tunnel, you have much lower speeds compared to the actual aircraft. So, therefore it is quite possible or it was anticipated by the people who did the experiment that this is a place approximately where the boundary layer will be transitioned to turbulent in the real aircraft. But, if we do not put this strip then the boundary layer will not or may not transit to the, may not become turbulent at that point. So, it is done intentionally, ok. One interesting anecdote is that the one of the authors of this paper M.A. Vaziri was my PhD colleague. I just saw it and I realized that he is author of this paper.

(Refer Slide Time: 9:05)



Ok. So, I have taken a small image of a paragraph from the paper and important thing is this- the location of the transition region has essential effects on the drag coefficient. We will study later on about drag coefficients. This location was controlled using trip strips or the wing leading edge. And at the nose, the size of the strips was such a small one, and figure four shows the position of the trips that is what I showed you ahead. So it is intentional, why? Because the Reynolds number of the aircraft would be 1.2 million.

Whereas, the Reynolds number in the wind tunnel, sorry in the wind tunnel 1.2 million and 13 million in the actual aircraft. So, it is almost 10 times more. So therefore the flow will not be the same if I do not create. Ok. So, what do we learn from here? That when you do wind tunnel testing and you want to simulate the real life conditions, you have to create transition at the place where you extract transition in the actual aerodynamic body. Otherwise the results will not be matching. Ok.

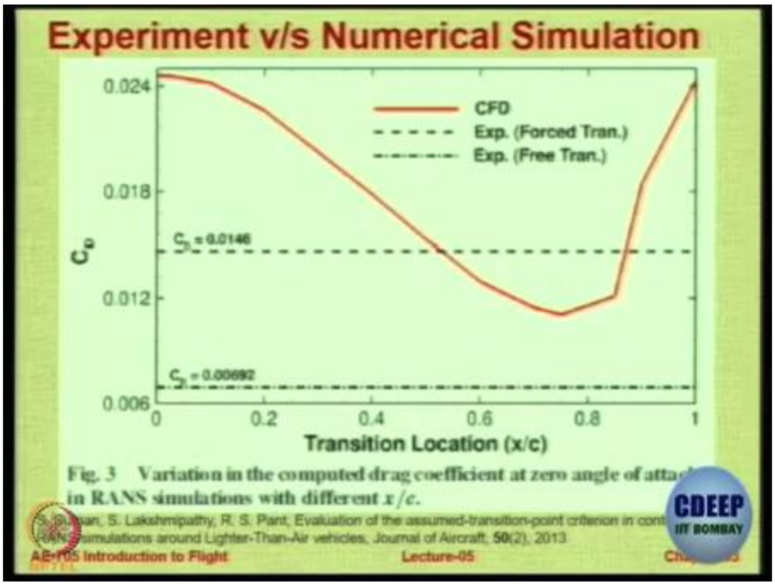
(Refer Slide Time: 10:15)

So, now sometimes there is a problem. So my understanding was that if you put a transition strip the transition will take place and the flow will become turbulent. So, recently we did a study, so what happens is, this is an . This is one particular photograph from another paper in which some studies over airships were done in a wind tunnel, ok.

This is by Wong Atol. It is an experimental investigation on a particular airship. So, what they have done is they have put three strips. One right in the nose, one in the middle of the envelope and one near the fins, because they want to create transition at these places. So, they have figured out that in the actual, this is a scaled model. In the actual airship there will be transition right in the beginning. So therefore they have put those strips and near the fins the flow gets disturbed anyway so there is a transition, the flow generally separates. And then after doing this they did wind tunnel testing, ok.

I am trying to slightly go ahead. I am sorry. I was debating whether to show this or not. Because I will talk about things like lift coefficient , drag coefficient which I have not taught yet. But I wanted to show you right now the effect of transition. So, this is again a picture from their, this is a picture from our paper. You can see I am one of the co-authors of this paper. So what we did is we tried to simulate this particular wind tunnel test numerically, ok.

(Refer Slide Time: 11:43)



So, the red line that you see is the value of a parameter called drag coefficient for various transition locations, specified intentionally by us. So in the numerical exercise or in the computer program which we have written you can specify transition point. It is not possible in all computer programs; especially if you look at commercial programs you may not be able to do this explicitly. But if you have your own program, you can, ok.

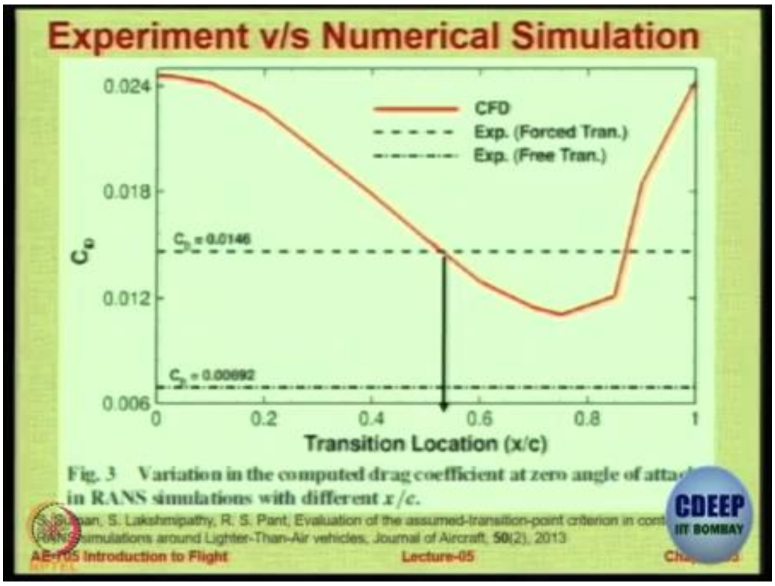
So, in this case Sunil Lakshmipathy the second author, he works in DLR in Germany and there is a code available with them. Very powerful code which was used for this study. You can specify transition. So what he did is he specified transition physically at 10 percent, 20 percent, 30 percent etcetera and calculated the value of drag coefficient for various transition points. So, the red line shows the captured or estimated value of the drag coefficient at various transition locations for the same geometry, same Reynolds number and the dotted lines indicate the value reported by the authors of this paper based on experiment. So, what you notice is that there are two results, free transition and force transition.

Force transitions means when you put a strip and you create transition. That is called as force transition. Free transition means you do not have any strips there, the flow will automatically become turbulent at some place depending on the critical Reynolds number, which is not known.

So, we observed that if you do not put strips the value of the drag coefficient you get is very very small 0.007 approximately. Which we are never able to capture. So even when we do the CFD analysis with the transition at zero percent or 100 percent that means fully laminar, fully turbulent or in between, nothing happens. So, we are unable to capture and this made us very much worried. Because we thought, why I am not able to capture it, then we said oh, because they have done transition.

Now the transition was forced right in the leading edge ok. So, therefore when I have transition somewhere here I should get the value of CD experimentally matching. Now my value is almost 0.024 but the value they have got is 0.0146 so we got worried initially. Because we thought they put a strip, the flow will become turbulent there and therefore the transition will be somewhere here. So, when we specify transition also at that point we should get a match but you can see there is a huge mismatch between the value that was reported by them and value that we got. So, this got us a bit worried in the beginning.

(Refer Slide Time: 15:07)

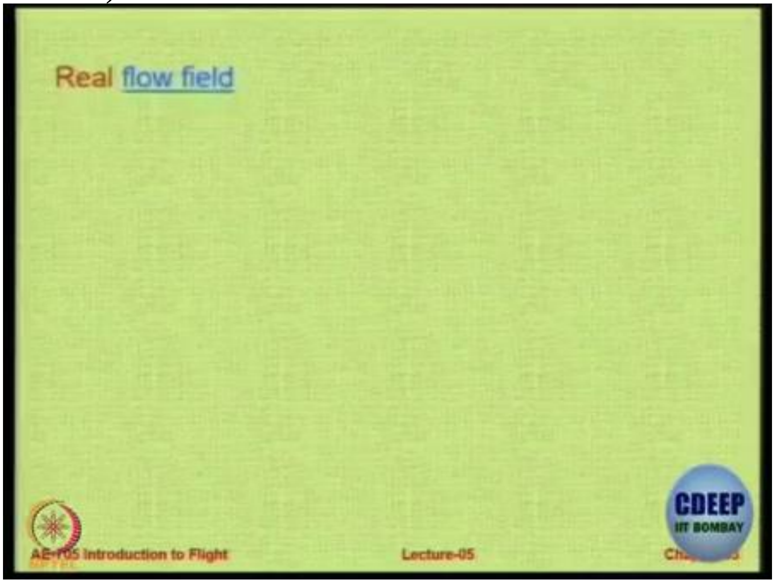


And then we realized that the shape of the body is such that there is a favorable pressure gradient in the front portion, because the curvature of the body is slowly changing. So, the pressure of the air, ambient air at this portion is continuously, you know it is more here and less here so the air will be pushed behind. So, then what we did is, we said ok, let us see where does it match. So, we found that we are able to recover this value when the transition is almost at around 55 percent and that is roughly here, ok.

So, basically, what we have seen here is that even if you put a strip it may not necessarily create transition. Another factor is the pressure gradient. If it is adverse it will trip very quickly. If it is favourable it may not, ok. So, you can see when you have a transition in the beginning the flow is, more flow is, so as you move transition point behind more and more flow is laminar and then remaining is turbulent so therefore the drag is reduced. When you come to the point where the separation occurs suddenly drag is increasing. And behind that the drag is more as more surface is exposed to separated flow, ok.

So, this is one of the results from our paper in which we put transition at point 6 of the length. And notice, we observe here that the flow at around 90 percent shows flow separation which is what they also reported in the in the paper. So, what have we learned from here basically is that if you put a strip for transition it should be a reasonably disturbing strip, just any strip may not help. And secondly, even when you put a disturbance or a strip transition may not occur because of the pressure gradient.

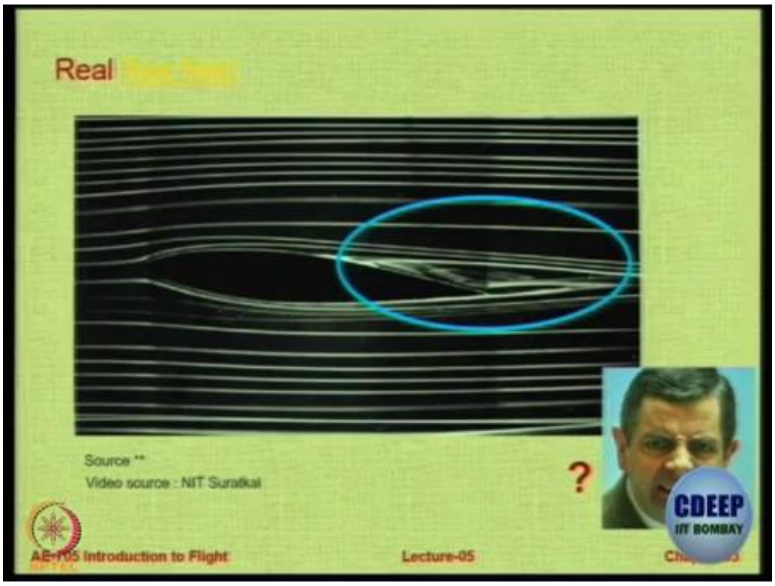
(Refer Slide Time: 17:34)



So, let us have a look at the real flow field with boundary layer. So we will just recap now. Now we are going into practical realities of laminar and turbulent flow. So here is a real flow field. So, this is an experiment by some students from NIT Surathkal. So, you can see this is a real flow field. Notice that there is a huge zone here where the flow is actually coming back. This is the recirculation zone. When you reduce the angle then it becomes much smoother and as you increase the angle the disturbed area behind the body becomes larger and then at some point you start seeing flow reversing direction. So, basically at this angle the flow from here is leading the surface. So this whole area has got separated flow.

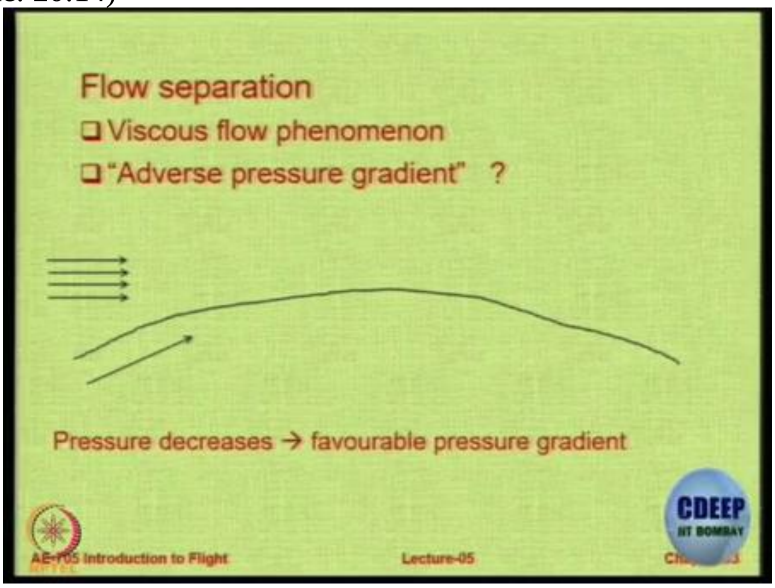
In other words the aerofoil here is not playing any part in the lift generation, ok. You bring it down slowly, and you will see that the flow starts getting slowly better and better. So, what are these? These are basically smoke trails. So, what are these? Are these pathlines, streamlines, streaklines? Think about it. Are they streamlines? Yes, they are streamlines. A streamline is a theoretical concept you will never see it in real life except in experiments like I did in the tutorial. You will never see a streamline of flow. It is something that you contrive for calculation. It is a theoretical line at which velocity is perpendicular every time, ok. It coincides with the pathline in steady flow that is a different thing but streamlines are not something which are normally visible, ok.

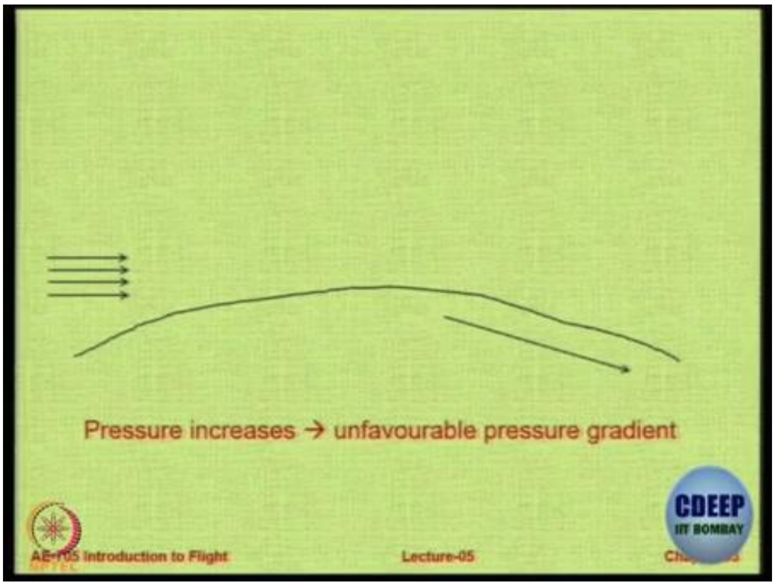
(Refer Slide Time: 19:39)



So this is the area where the flow is separated. So, earlier I showed you viscous, non-viscous or inviscid flow over a plate or a laminar flow over the same kind of aerofoil. It was very smooth and joining. Now we see there is a disturbance here. The angle is not very high but still you have disturbance this is because of the presence of viscosity. It is because of the Reynolds number, ok. So this is what is really of concern to us because that area is getting disturbed. So this is the effect of viscous flow.

(Refer Slide Time: 20:14)

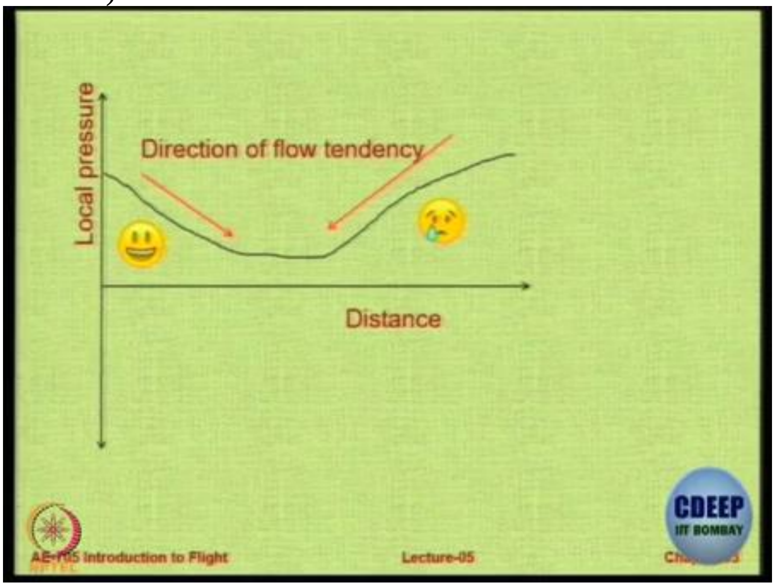


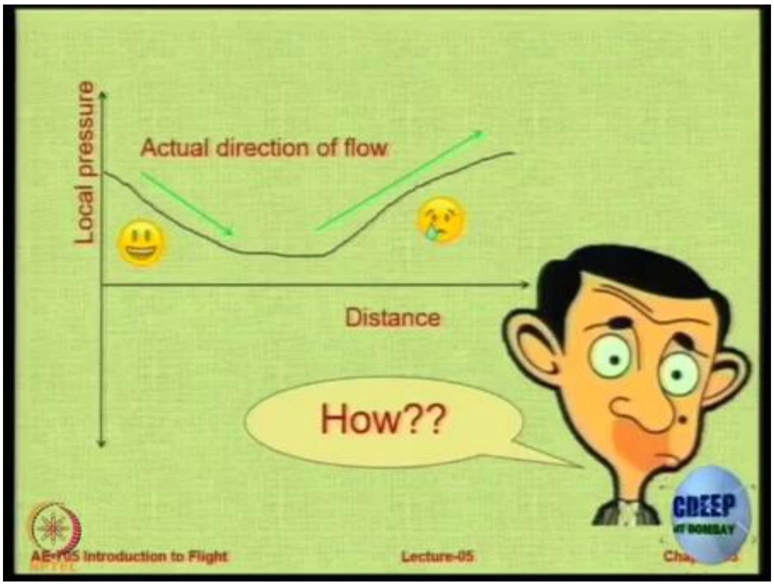


The effect is flow separation. It is a viscous flow phenomenon. So, if your flow is inviscid you will never get separation, ok. And that is the concept of adverse pressure gradient which causes separation or which triggers separation. This is what I showed you in the previous experiment. So, if you have a body like this and if you have a flow you can see that the pressure decreases as you go ahead. That means the pressure is slowly reducing. In other words, if the pressure reduces there is a tendency of the flow to go. It helps in the flow, ok.

And at that trailing part of the body the pressure is going to actually increase with length. So, that means there is a back pressure which is trying to oppose the flow. Therefore there is a chance that there will be separation.

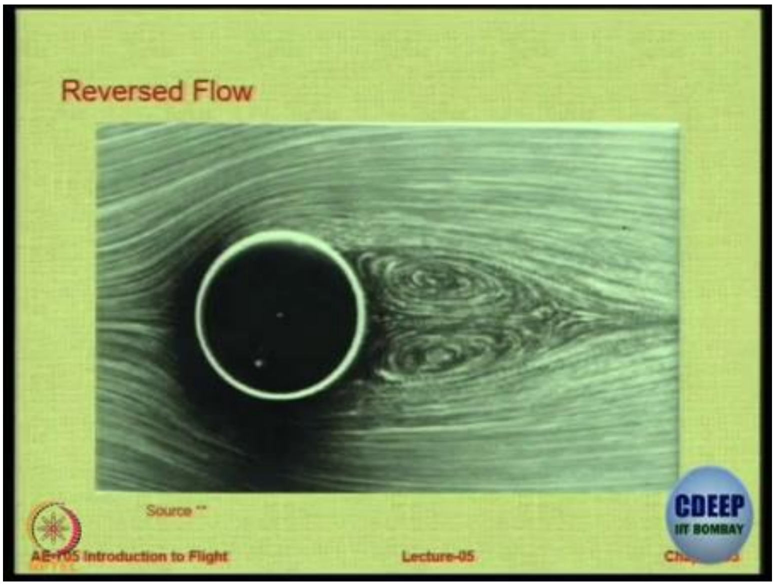
(Refer Slide Time: 21:07)

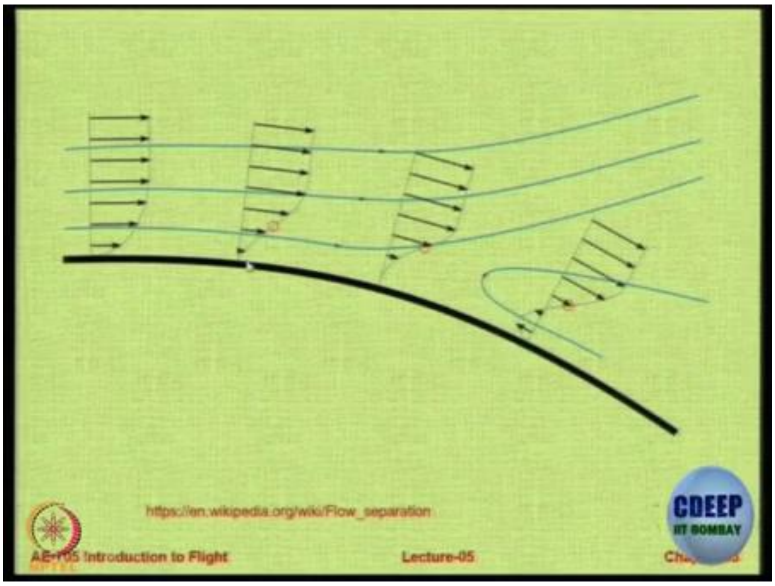




So, the local pressure, it is good if it decreases it is good, but after some distance there is a problem and that causes separation, ok. So, let us see why and how this happens. So, because there is momentum in the flow and that, this momentum of the flow is actually going against the pressure. So, if the pressure is favorable or the pressure gradient is favorable it is actually pushing the flow. It is like helping, go. But when you are going against the pressure gradient where the, it is the gradient is positive then it is actually opposing the flow. So the momentum of the fluid faces opposition and if the momentum of the fluid is not enough it will get disturbed, ok. So, that is the problem. So, let us see what happens now, ok. What is happening in the separated flow region?

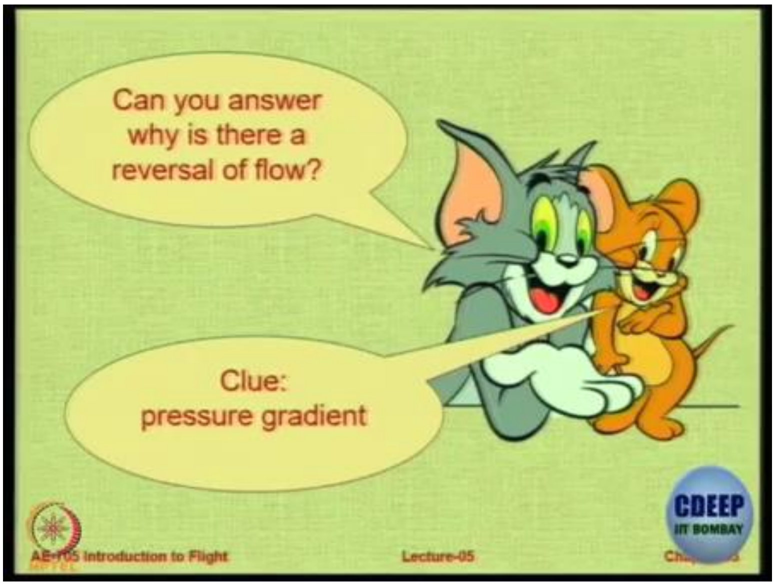
(Refer Slide Time: 22:13)





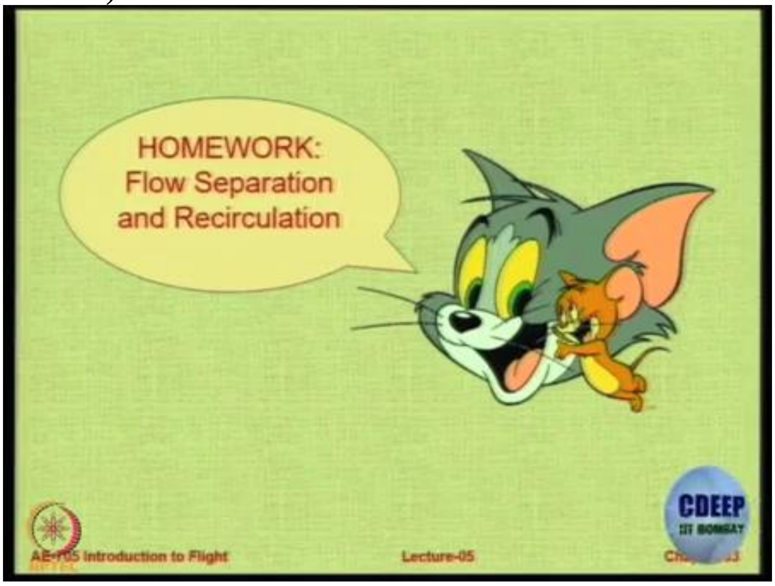
You have this reverse flow which I have already shown you in the video. The area where the flow is coming against. So, larger the area of reverse flow larger is going to be the resistance to the motion. So here is a sketch along a particular, along the length of a shape of a body. We are plotting the boundary layer. So you can notice that at this place you start facing problems, you can have a situation like this where the flow velocity is positive but in this area it is negative. That means it is the reverse flow region and this is what causes separation.

(Refer Slide Time: 22:54)



So, in a well-designed body we should avoid creating a shape or a flow structure where the flow is getting reversed because that means the body is not playing. So, the best way to avoid reverse flow is to control the pressure gradient. You can do it naturally, you can do it artificially, but you should do it. So that if the pressure gradient is too much adverse you will not be able to, you will not be able to control and the flow will separate, ok.

(Refer Slide Time: 23:29)



So, now you have some homework which is on the Moodle page. You have to give examples of flow separation and recirculation. You have to locate information about why it happens and, more important is how do we control? So, I am looking for some information regarding- if there is flow separation what is being suggested to control the flow separation. How do you create favorable pressure gradient in the region where the flow is separating? So, you can think of giving examples of here is a flow, it is separated. This is what we do to recapture that flow or avoid the separation. That is your homework that happens on the Moodle page.

(Refer Slide Time: 24:16)

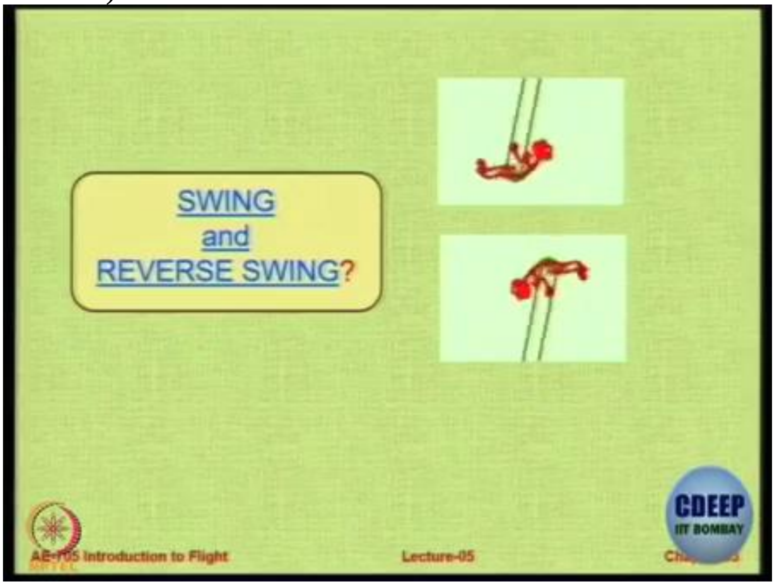
Second thing is, is there a difference in the way separation mechanism works in laminar boundary layer as compared to the turbulent boundary layer? They are not the same. So what is the difference? That is also something which I am leaving it for you, ok. Because, the turbulent boundary layer basically delays the separation. Is very interesting, the turbulent flow leads to higher resistance on higher drag but the turbulent flow actually pushes the area of separation behind, because it energizes the flow, ok. I think we have had enough of theory. So, now we take a break and go into something more interesting but still in fluid mechanics.

(Refer Slide Time: 26:10)

We will watch how fluid mechanics is used in two sports. One of them is cricket so let us just watch a small clip on how the fast bowler, now you will see three balls in this particular video clip and let me put the audio also. One of the best players. This is the first ball. But it is slow motion. See how the ball is curling inside. Now there is no doubt. Let us see there is a slow motion. See, how the ball curls inside.

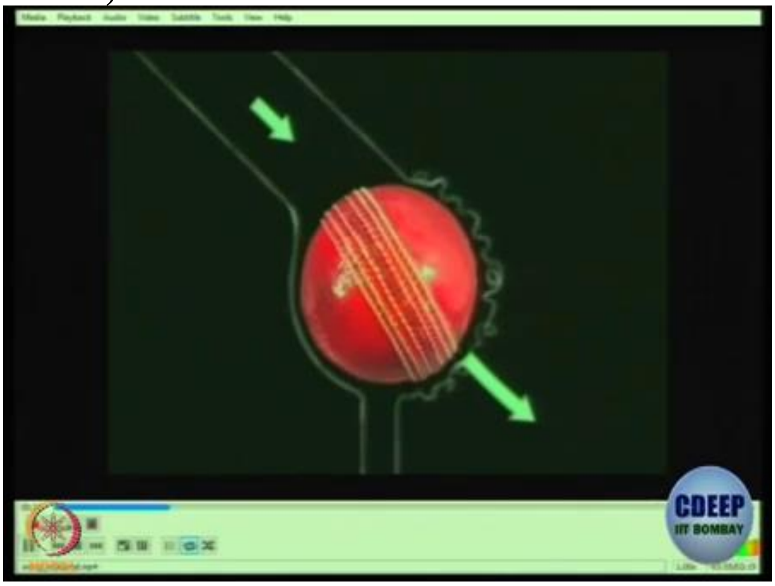
Ok. So, we saw three balls bowled by the bowler, all three of them the batsman could not handle and the third one obviously the wicket is down. So, this particular phenomenon is called as a swing in which the ball is curling towards one side.

(Refer Slide Time: 27:13)



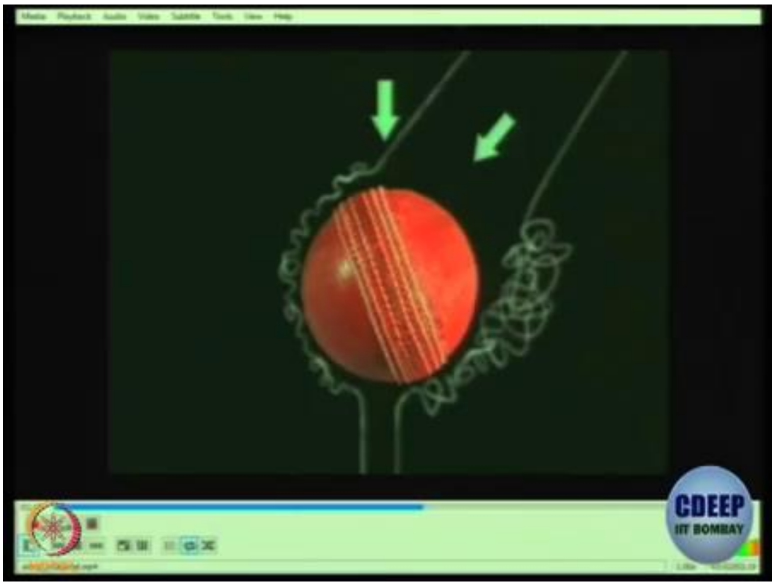
So, we have swing as well as reverse swing, both are available. So, let us see what is this phenomenon and how it is done.

(Refer Slide Time: 27:26)



So one side is rough, the bottom side is smooth. So, it swings towards the smooth side. Swinging towards the smooth side.

(Refer Slide Time: 28:19)

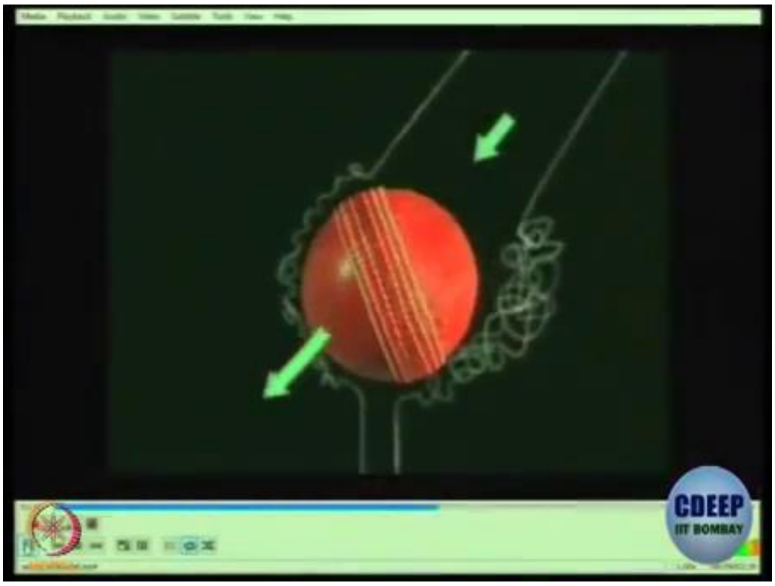


Video: Changing the direction of the seam alters the way the ball will swing. In the 1970s the great Pakistani captain Imran khan discovered he can turn his science on its head. Imran found that he could swing a ball in reverse. The reverse swing happens when a ball swings in the opposite direction to conventional swing. The ball is travelling in this direction, air is turbulent around both sides of the ball, but when it reaches the seam it is tripped and becomes even thicker. This thick layer peels away from the ball earlier putting pressure on this side of the…

Professor: So, in the normal swing balling one side is smooth, the other side is rough, okay that is why the players keep polishing one side continuously, ok. They want to keep one side smooth, the other side is getting rough because of playing. This is the beginning of the innings when there are few overs spent. So then, when you have the ball you throw at a particular angle, you hold the seam and throw it. So the seam is tripping the boundary layer. On one side it is laminar, on the other side it is turbulent and therefore, there is a tendency of the ball to move towards the shining side. Now this is reverse swing, this happens when the ball is 60-70 overs old.

It is rough on both sides. So, on both sides we have turbulent boundary layer. But, because of the seam on one side the boundary layer is tripped early. So therefore, that area is going to, so now what will happen is now it will actually swing towards the rough side.

(Refer Slide Time: 29:49)



Video: So, the ball swings towards the left. But, what no one really understands is why the ball swings more during some matches than others. Rod Marsh: the time it swings most as far as I am concerned is, is when there is this apparent bed of humidity over the pitch that is to me when the ball really does swing more than other times. Now, why this happens? I do not know but I just know it happens as long as I can recognize, the fact that it is going to happen as a player or as a coach, then I am quite happy with that.

Professor: Basically, he is a wicketkeeper; he does not know why it happens.

(Refer Slide Time: 30:25)

Video: Perhaps the most mysterious of bowling arts is spin. Australian Shane Warne is a master.

Professor: Now this is something else. This is not swing, this is spin. So, that is why we skipped. So, the bowlers also use fluid mechanics to give the ball a variation. Now, I want to show one of the best starts between India and Pakistan in which our bowler used swing bowling to take three wickets in three balls, ok. The first three balls:

(Refer Slide Time: 31:00)

Professor: This is the first ball. Look at the deviation. The safest player in the slips is Rahul Dravid. Second ball, look at the speed of the ball. The ball is 76 miles per hour. It is a beginning of the match, first over. See how it is swinging.

Look at the ball, it is shiny outside. I do not think there is a slow motion after that. Ok. Let us move to another game called as golf. Ok. Yeah, that is right, that is why they rub. They rub the ball because they want to make it smooth on one side. That is the reason why they rub the ball. Ok. Golf is another game where they use fluid mechanics by dimpling the ball. So, when you dimple the ball you ensure that the flow is turbulent. And, when the flow is turbulent then the boundary layer is more stable. The flow separation is delayed. And because of that the length to which this particular ball will go is nearly twice compared to a smooth golf ball.

So, let us see why we have these dimples. And, now the diameter of these dimples, their depth is also very carefully designed. It should be sufficiently deep so that it causes disturbance but not separation. And the number of dimples and the dia of the dimples and the shapes are also carefully monitored. As we can see in this nice video this shows you the working of the dimples on the golf ball.

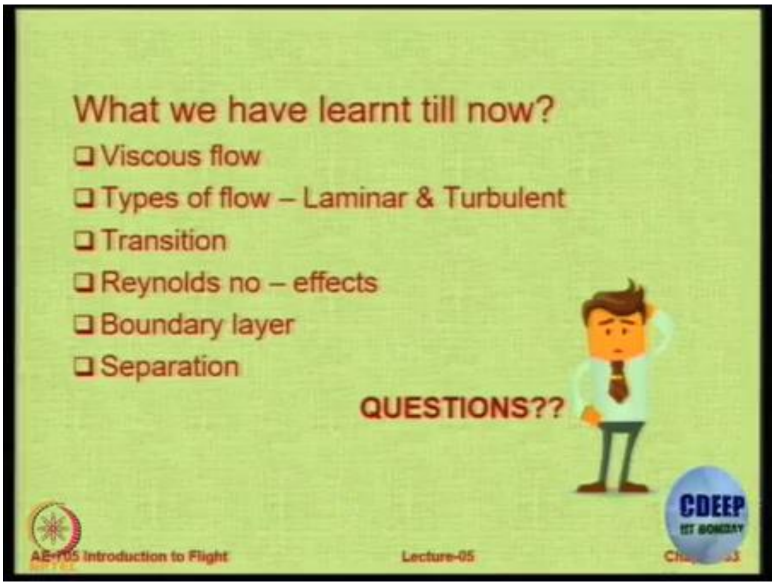
(Refer Slide Time: 33:59)

Video: It is the golf ball so unique. It is not just its small shape, but hundreds of small impressions or dimples on its surface. Golf ball has dimples to reduce wind resistance or aerodynamic drag. When you reduce it you can make golf balls going out a lot farther. Adding dimples to the ball changes how the air flows over it. As the air travels over one of the dimples a tiny pocket of turbulence or air disturbance is created on the surface. It tries to go in, and then pass a region where it is actually detached but then by the time you get to the next dimple on the ball it reattaches itself and in the process of that detachment reattachment that is what creates the turbulence.

Instead of impending the flight of the ball, these tiny pockets of turbulence allow the closer layer of air to travel tighter around it. A more attached air flow creates a smaller wave and thus a smaller low pressure zone which means less drag. Even the slight change can make a big difference. A golf ball with dimples will go almost twice as far as a golf ball without. Well dimple sizes, shapes may differ. They remain a crucial aspect of all golf ball designs.

Professor: Ok. They also use curling of the flow. So, if they have to go past an obstacle they actually impart spin to the ball. Just like we saw in the Coanda effect earlier.

(Refer Slide Time: 35:37)



Ok, so summing up what have we learned till now? What have we learnt? Number one, we have learned that there is something called as a viscous flow which is a natural property of any flow. In real life all flows are viscous but the viscosity may vary. As we saw in the beginning the race among various fluid particles, we saw that honey has the highest viscosity, so it took the maximum time more than 20 seconds to come as against 0.4 seconds for water. Ok. So that is because the viscosity for honey is far, far more than water. And color of the fluid does not matter. It is the viscosity that really tells you how much friction will be created.

So viscous flow is reality. There are two basic types of flow, laminar and turbulent depending on the value of the Reynolds number of the flow. Laminar flow is the one where the drag is less because there is less mixing between the layers. It is a very smooth flow but laminar flow can be maintained only when the flow Reynolds number is below a critical number. And also in the next class we will study about flow separation in more detail. We will see about the stability of the laminar and turbulent flow, so laminar flow has less drag but it is more unstable.

Turbulent flow on the other hand is the one where there is mixing happening between the various layers. Turbulent flow will have more drag than laminar flow but the turbulent flow as you saw in the golf ball, the turbulent flow actually reduces the weight behind the body and hence it reduces the adverse pressure and hence it gives you lower drag. Plus it does not allow the boundary layer to separate easily. So turbulent, turbulent flow is more stable. Separation is less in turbulent flow than in laminar flow.

So, between these two we have transition. Transition does not depend upon the shape of the body. It does not depend on the angle at which the body is placed, it depends on the property of the fluid called as viscosity. So, and the parameter called as the Reynolds number which is the ratio of the two forces inertial and viscous forces. When the Reynolds number is beyond a number called as a critical Reynolds number which depends on temperature, viscosity, surface roughness, many other things which we have not discussed so far but the moment you have a transition the flow will become turbulent. Ok.

Then, Reynolds number is the parameter which is used to identify the transition location. We also saw that there is a small area near the surface of a body called as a boundary layer in which the viscous effects of the boundary layer are limited. So, therefore if you have a non-viscous calculation procedure you can calculate the forces acting on the body by assuming the body of a shape equal to the body plus boundary layer. A new body which is body plus boundary layer. If you investigate non-viscous flow on that you can get pretty much most of the values. Ok. Except you will not get drag because drag depends on the viscosity a lot.

So the boundary layer is a small area which is where the viscous effects are confined. The boundary layered is zero at the beginning of the body, it slowly increases and then when there is a transition the boundary layer becomes very thick. So a laminar boundary layer is thinner and a turbulent boundary layer is normally thicker. And when you have a separation when you have a transition, when you have a transition from laminar to turbulent there is a tendency in the end that you might reach a place where there is adverse pressure gradient or where the flow velocity is reversed, so that area is called as separation. It has to be minimized by careful design because when you have a separation it means the fluid particles they are not taking part in generating the forces except the drag force. So the area of separation has to be kept as low as possible under the operating conditions.

And now, are there any questions before we wind up for the day? Do you have any questions based on what we have seen so far? Yes.

Student: Sir, in the video which you show of the hat-trick, sir the ball was new so it must be smooth from both the sides, right?

Professor: Yes, that is true. But, by holding the seam in a particular location when you throw it, the seam will still trip the boundary layer. So in a brand new ball also when both sides are smooth you can still create swing because you can use the seam position to trip. And that will create a difference in both the sides. But that becomes more. So what people do is when you start playing, the ball gets disturbed from both sides. So, intentionally one side is chosen by the bowler and they keep spitting on it, they keep rubbing it because they want to improve. Because, the seam of the ball also gets broken and disturbed with time.

So you are right, swing will take place even in the brand new ball. Even in the first ball of the innings also it can be swung. In the recent match between India and Pakistan you must have seen Rohit Sharma, he was not able to face Mohammad Amir's bowling because it was swinging right in the first. So the third ball he was out or the second ball he was out. So, it was a beautiful swing bowling. So swing can take place even because of the seam. Yes, any more questions? Yes, take the mic please.

Student: Sir, in that example of airplane wing where we have the trip strip on the wing at the last of the wing, so you said that we intentionally make these trips because we want turbulence flow after that strip. So, my question is why we want turbulence flow after that?

Professor: See, when you do wind tunnel testing or any numerical analysis you would like to capture the flow phenomenon as it exists on the actual aircraft. Correct? That is the purpose of wind tunnel testing to predict the flow parameters of the actual aircraft. So that means the conditions to which the actual flow is subjected we want to replicate in the laboratory. So the actual wing is going to fly at a very high mach number. Let us say 0.9. So wind tunnel will have only Mach number of nearly 0.2, 0.3. Ok.

The actual wing will be very large in size. You can have only a scaled model. So we know that if we want to get the correct flow characteristics we must maintain the Mach number and Reynolds number. Mach number we cannot maintain. So we will say okay let us try to keep the same Reynolds number. So your model is let us say ten times smaller, so Reynolds number is one-tenth already. Let us say your speed is ten times less, so you have hundred times lower Reynolds number. So if I do not do anything, I just put the scaled model of aircraft in the wind tunnel I will operate at a Reynolds number of approximately hundred times lower than the actual. If the ratio of the Mach numbers and the size is one-tenth, correct?

So therefore, we know that as Reynolds number changes or I should say that Reynolds number is a very important parameter affecting the flow behavior. So in an actual aircraft because of large Reynolds number it was 13 million in this example. When you have 13 million Reynolds number then in the actual aircraft the flow is going to become turbulent at ten percent of the wing leading edge. This is an observation. But when I put a model in the wind tunnel because Reynolds number is hundred times less it may not trip at 10 percent, it may trip at 30 percent.

So then I will not have the correct value of drag. So what I do is I intentionally create transition at the place where it will be on the big aircraft so that at least my experimental data is not wrong. That is the purpose why we use transition strips.

But as I showed you it does not always work. As I showed you in the next example they put a transition strip but the flow did not separate. The flow did not become turbulent there. The flow became turbulent only at around 55 percent. Yes?

Student: Sir, generally this experimental test can be carried out from the model which is made first and the real aircraft will be made at the end of this exercise. So how do we have a prior information that where it should trip in the actual aircraft?

Professor: Very good question. We do not have. So what we do is in most practical aircraft the transition will take place right in the leading edge. We assume that the wing will be in fully turbulent flow. So what we do is we put a strip in the leading edge, so there is no chance of the boundary layer being laminar. So in typical aircraft design exercises the fuselage will have a laminar flow only about 5 percent. Never 5 or 10 percent. There are some exceptions.

There are some aircraft where the shaping of the fuselage is such that the transition starts at 10-15 percent but generally for any because there will be an imperfections in fabrication. There will be maybe there is some rivets which are protruding out. So therefore we assume that right in the front there will be transition to turbulent. Okay for the wing a very well designed wing 10 percent, 15 percent we could think it to be laminar, after that you will have items on the wing which will protrude, maybe there is de-icing pad or there is some projection on the top. Maybe there are slats which have a small edge, so the flow will become turbulent.

Ok, so therefore, in normal wind tunnel testing now, here this testing was done after the aircraft was designed. Because here they are trying to change the rear of the wing to reduce the base drag, so therefore they have. They have an idea what is the location of the probable location of the boundary layer transition and that is what they are simulating. So what you say is right. In the actual aircraft when we design it normally we assume 100 percent transition. Ok. What is your name?

Student: My name is Srinath.

Professor: Yes.

Student: The laminar flow changes to turbulent flow when Reynolds number crosses critical Reynolds number.

Professor: Not only that. Not only that, it changes to turbulent flow at a critical Reynolds number if everything else is remaining same. But I can do it earlier by doing some nasty things. Ok.

Student: What is the main factor which influences this? I mean what is the factor which converts laminar flow to turbulent flow?

Professor: The Reynolds number. Because basically the transition takes place because of the interplay between the inertial and viscous forces where the viscous forces become predominant.

Student: Sir, for viscous forces to become predominant is it necessary for the ratio to be in millions or lakhs?

Professor: No, no not necessary. For example, in pipe it is 2900. It is not. As I said the critical Reynolds number is not, it should not be only in millions or in some range. It varies. See I have not talked about things like temperature, surface roughness, they also affect. So predicting transition is a very big challenge in fluid mechanics even today. It is very difficult to predict transition on any general body. Okay. So many people are able to only do…so for example there are some models. There is some model called as E-power N transition prediction model, so these are all empirical models. They, they are based on data or studies in the past about flows where transition has been observed.

So it is like mix and match kind of a thing. So even today there is no reliable method or code available which will say okay if this is the flow if this is the geometry if this is the property 100 percent transition will take place here, it is not possible. Okay. But for a simple thing just like a pipe Reynolds observed that when this number goes beyond 2900 or some value the flow becomes turbulent, but that is only for a pipe flow with a simple fluid uniform steady flow. Okay but for a really complicated shape it is difficult to actually predict the transition point.