Transport Phenomena.
Professor Sunando Dasgupta.
Department of Chemical Engineering.
Indian Institute of Technology, Kharagpur.
Lecture-27.
Turbulent Boundary Layers (continued).

In the last class we were working on a problem in which there was a horizontal duct and air is sucked into the horizontal duct. So outside of the horizontal duct it is atmospheric air, the pressure is equal to 1 atmosphere. And there must be a suction created downstream for downstream into the duct, which would cause the air to come from outside into the duct. It has been mentioned that at the entry point which is well rounded, the velocity of the air entering the duct is provided equal to 10 metre per second.

And at a point slightly downstream from the from the location one, that is the entry of the duct, the thickness of the boundary layer has been measured and it has been also mentioned that the flow turbulent from the very beginning. And we were asked to and the profile of velocity inside the turbulent boundary layer is following the 1/7 power law. So there were 3 parts of the question, the 1st part we have to show that delta Star which is the displacement thickness is related to delta, the thickness of the boundary layer at a given location, they are related by delta Star equals delta by 8.

In the 2nd part it has been asked that find out what is the pressure, gauge pressure at location 1 and at the location 2. So the way we have handled the problem is, since the velocity profile is known to us, using the definition of displacement thickness we should be able to evaluate what is the value of displacement thickness at location 2. Now this is important because in order to obtain the pressure difference between location 1 and location 2, we intend to use the Bernoulli's equation. We also realise that Bernoulli's equation is valid only for inviscid flow.

If viscous forces are present, that will cause a reduction in the pressure as a result of friction of the fluid with the solid surface. The way the frictional effects are included in Bernoulli's equation is by the head loss, where head loss denoted normally by H is related to the friction factor and other parameters. But since we do not know for this problem how frictional effects are to be included in Bernoulli's equation, then we would like to convert situation into an inviscid flow problem.

In order to use inviscid flow, we must ensure that we are only considering the core region of the flow in which the velocities are equal to the free stream velocity but without the effect of friction losses in between 1 and 2. The way to do that, if you recall the definition of displacement thickness, it is the thickness by, it is the distance by the platform has to be raised in an inviscid flow to obtain the same pressure drop. So between 1 and 2 there would be pressure drop due to viscosity, due to friction, due to liquid, due to fluid friction.

However if we want to use Bernoulli's equation and convert this to an inviscid flow situation, we need to raise the platform by a distance equal to the displacement thickness. Once we do that, then the flow area at location 2 is going to be constricted in comparison to location 1, however the Bernoulli's equation can now be used, because by definition of displacement thickness, the flow, the flow entirely is going to be in the inviscid flow region. So at location 1 we understand that the thickness of the boundary layer is 0, the thickness of the displacement, the displacement thickness is also 0.

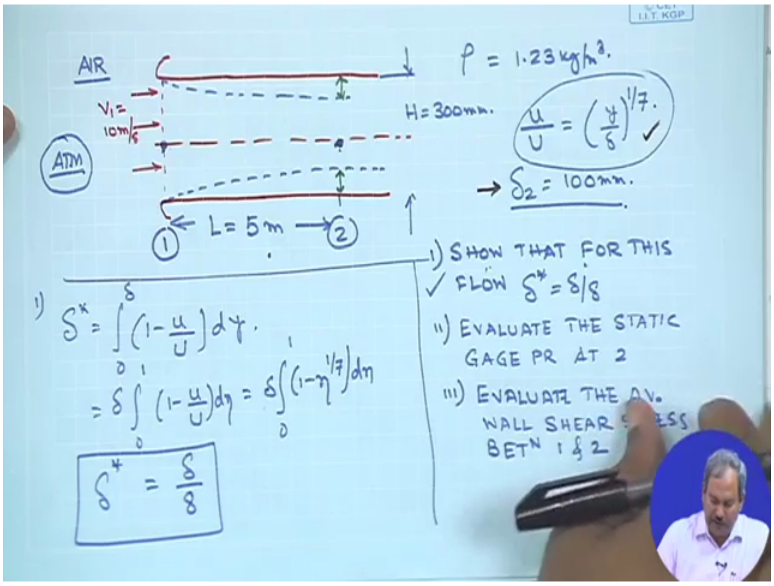
But at location 2, since my profile, velocity profile inside the boundary layer is given and in the 1st part of the problem I have a relation between delta Star, the displacement thickness with delta, then I should be able to compute what is delta Star at location 2. So if I constrict the area from the top and from the bottom by a distance equal to delta Star, then whatever be the flow along the streamline, Bernoulli's equation in its inviscid form can be used.

So we are going to use the concept of displacement thickness, the Bernoulli's equation and equation of continuity in order to obtain the pressure at 1 and pressure at 2. So that is what we have done in the previous class, I will very quickly show you the derivation, the solution once again but the 3rd part of the problem is more interesting. In the 3rd part of the problem we were asked to calculate what is going to be the shear stress, average shear stress between 1 and 2 on the control volume of fluid that is flowing through the duct.

The physical property of the fluid, air in this case is provided, so we 1st need to calculate, we 1st need to show or derive the relation between delta Star and delta and secondly using Bernoulli's equation in an inviscid flow situation find out what is the pressure, the gauge pressure at 1 and at 2. We understand from the beginning, since outside of the duct, the pressure is atmospheric, so at the entry point, the pressure must be lower then the atmospheric pressure and at location 2 it will be even lower as compared to 1 and obviously as compared to the pressure outside.

So if I if I define, if I try to find out the gauge pressure at 1, it is going to be negative since the pressure at that point would be less than the atmospheric pressure and at location 2, it is going to be even more negative such that a pressure gradient between point 1 and point 2 would exist that would cause the fluid to move flow from 1 to 2. So the way we handle the 1st part of the problem, the relation between delta Star and delta in our previous class we would prefer to that now.

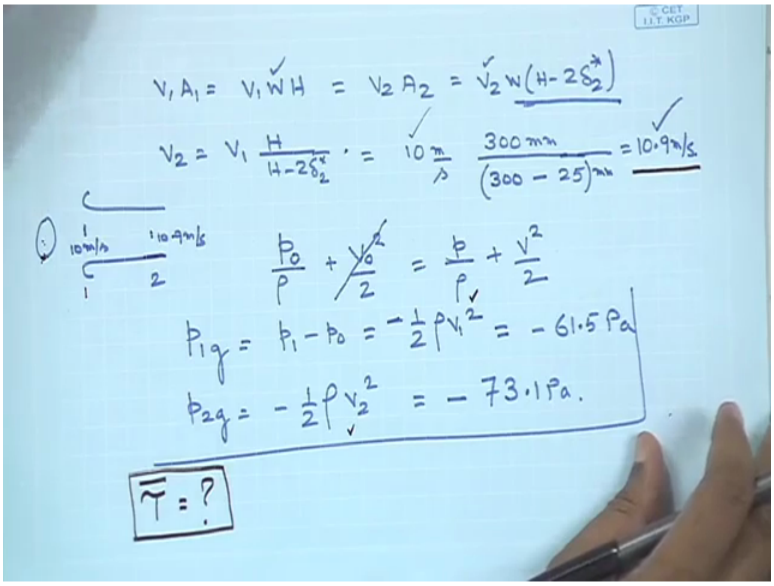
(Refer Slide Time: 7:04)



So this is the figure of the problem where we have air at atmospheric pressure, the entry point, the velocity was given as 10 metre per second and at location 2, the thickness of the boundary layer was given as 100 millimetre. And it has been it has been given in the problem that the flow inside the velocity, flow velocity u inside the boundary layer is related to the free stream velocity and the distance from the wall denoted by Y and delta which is the boundary layer thickness at that point and through the use of 1/7 power law which we know that is quite good in terms of fitting the experimental data in turbulent flow.

So the 3 things that we have to calculate is 1st show this relation, that delta Star will be equal to delta by 8, evaluate the static gauge pressure that location 2 and finally the average wall shear stress between 1 and 2. So we started with the started with the definition of displacement thickness which is 1 - u by U integration from 0 to delta, it is beyond delta u is equal to capital U and therefore there would not be any contribution of this integration for a value of delta for a value of Y greater than delta.

(Refer Slide Time: 8:47)



So we plug-in the expression of u by U from here and we can simply obtain the relation that is provided, that is we have the need to prove that delta star is equal to delta by 8. In the 2nd part of the problem we were to calculate what is the, what is the gauge pressure, 1st we use the equation of continuity where V1 A1 must be equal to V2 A2. What is V1 in this case, it is a width of the duct, it is a rectangular duct, width of the duct multiplied by the H at the entrance of the duct which is which is mentioned to be equal to 300 millimetres.

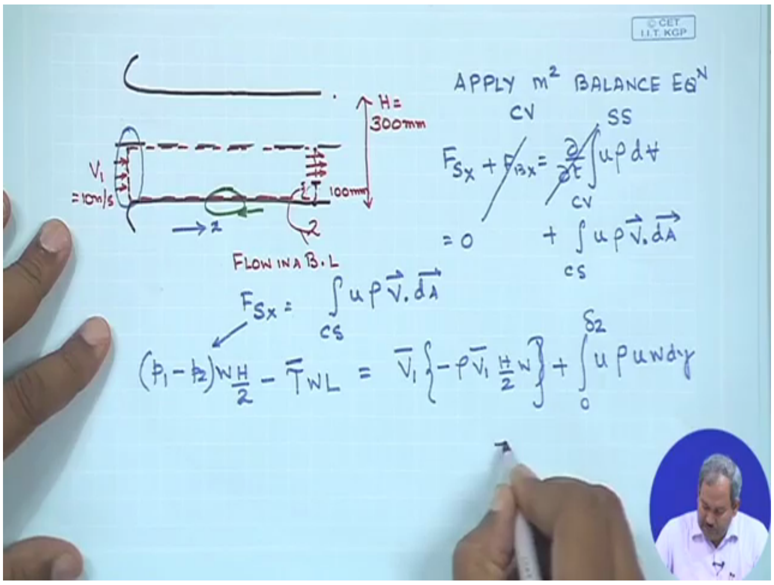
When we go to A2 in an inviscid flow, if we consider inviscid flow, the flow, the distance between the plates is now reduced by an amount equal to twice delta star. Delta star from the top and delta Star from the bottom makes the flow area to be equal to W which remains unchanged times H - 2 delta star. So V2, the unknown in this case is simply going to be V1 divided by a ratio of the area. When you plug in the values, you would see that the velocity at location 2 is equal to 10.9 metre per second.

So in this figure I have atmospheric air which has a velocity equal to 0, at location 1 the velocity is equal to 10 metre per second and at location 2, the free stream velocity, the velocity in inviscid flow equivalent inviscid flow is equal to 10.9 metre per second. Now I am going to write Bernoulli's equation between a point outside of the duct where the velocity is 0 and the pressure is equal to atmospheric pressure and between 1, once I write the equation between atmosphere and location 1 and this point and location 2.

So if I write this equation, p0 is the atmospheric pressure, V0 is a velocity of the air in the outside which is 0 in this case, P is the pressure at location, any location, it could be 1 or 2, and V is the velocity at location 1 or at location 2, so P1G, the pressure at 1, the gauge pressure at 1 would simply be the absolute pressure at 1 - the atmospheric pressure and from this relation you can clearly see this is going to be - half rho V1 square. And when you intend to calculate the pressure, the gauge pressure at 2, simply V1 to be substituted by V2 in an inviscid flow case.

So V1 is 10 metre per second and V2 is 10.9 metre per second giving you the gauge pressure to be equal to - 61.5 Pascals and P2 G to be equal to - 73.1 Pascal. So we would observe that there for P1 G is greater, greater than P2 G and therefore due to this pressure gradient there would be flow between location 1 and location 2. This is what we have done in the last class. Now only the 3rd point that is remaining is find out what is the average value of shear stress between location 1 and location 2.

(Refer Slide Time: 12:13)



In order to obtain the value of the average shear stress, wall shear stress between the entrance and location 2, I would draw the figure 1 once again, which is, this is the top plate and here we have the bottom plate, this is the centreline and I would take only half, I would only take the half of the duct and when we come to location 2, this is what it looks like. So over here, upto a distance of, so this entire thing is 300 millimetre, H is equal to 300 millimetre and at location 2, 100 millimetre is the thickness of the boundary layer.

So in here I am going to have boundary layer flow in a boundary layer of thickness delta 2 and over here the velocity is going to be constant. So since we have symmetry in the top half and in the bottom half, I have the velocity, average velocity to be equal to 10 metre per second which enters here carrying some momentum with it and the amount of fluid which leaves can be can be thought of as consisting of 2 parts, one which is a flow inside the boundary layer, the other, the 2 is what is the flow outside of the boundary layer.

So if I apply momentum balance, if I apply momentum balance equation on this CV, the control volume is this much, then what I am going to have is that F SX, this is my X direction + FBX is equal to dell dell T control volume u the velocity rho dV +, then I am going to have through the control surfaces there would be u rho V bar dot da, it is the area vector. And we realise that for this horizontal the case, FBX is 0 and this is 0 since it is a steady-state problem.

So I have then my reduced equation as FSX and 2 equals CS u rho V dot da. Now this FX has 2 components, one is pressure. So some, there would be a pressure, so the surface force, the pressure force which is forcing, which is acting on the control volume and the pressure which is acting against whatever be the pressure at 1 - the question at 2 multiplied by the area where the width is H but at location this would simply be equals H by 2. The pressure is a function only of X and we will assume that the pressure does not vary with Y, so the force, surface force due to pressure acting on the control volume would simply be pressure difference multiplied by the area.

And on this part, on over here, at this location, there would be a wall shear stress which is acting in the reverse direction which is causing the flow to slowdown which is the opposing force in this case, that is why we have this as Tao, average, W, the area would be W times L where L is the total length between location 1 and 2. So this Tao bar is the one which we have to evaluate and this is simply the wall shear stress.

When I go to the right-hand side, I am going to have some momentum which enters into the control volume through this point and which can simply be equal to the velocity multiplied by the flow rate which is rho V1 H by 2 times W. So if you consider the portion inside the bracket, this is nothing but the mass flow rate of the fluid, air in this case, mass flow rate of air which is entering into the control volume. Since mass is entering into the control volume, it is going to have a negative sign according to the convention that we are using so far.

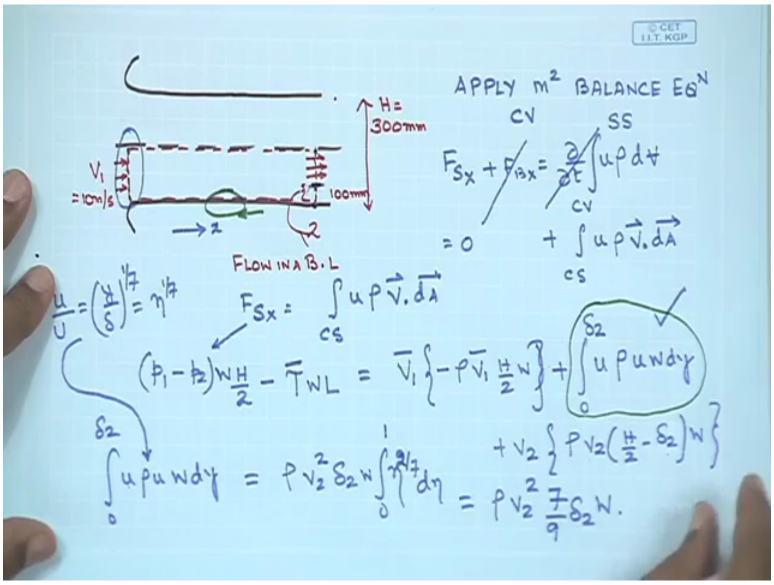
So this bracketed portion is the mass flow rate of air entering the control volume and the corresponding momentum associated with this amount of air is simply multiplied, multiply this with the value of velocity which is 10 metre per second at 1. And when we go into the 2^{nd}

part of the problem, there would be some mass flow inside the boundary layer and some mass flow outside of the boundary layer. So the 2nd term, that is the momentum going out of the control volume will have 2 parts, the 1st part being this up to 100 millimetre.

And beyond 100 millimetre, all the way up to 150 millimetre. So if you do that, the 1st part, 0 to delta 2, where delta 2 is the thickness of the boundary at location 2 u rho u times W dY. Look at it carefully, the 1st part, u rho u WdY, we have integrated over the entire thickness of the boundary layer would simply give you the mass flow rate. The mass flow rate of the fluid which is moving out of the come out of the control volume between the thickness starting at 0, between a point starting at 0, all the way up to 100 millimetre, that is the thickness of the boundary layer at that point.

And inside the boundary layer, the velocity varies as a function of Y and we know what is the variation of velocity which is simply going to be the one 7th, which will simply follow one 7th power law. And there would be another part which is outside of the boundary layer which will vary from H by 2 - delta that will have the flow which is moving out of the control volume at a constant velocity. So I will I will write the 2nd part as well.

(Refer Slide Time: 19:32)

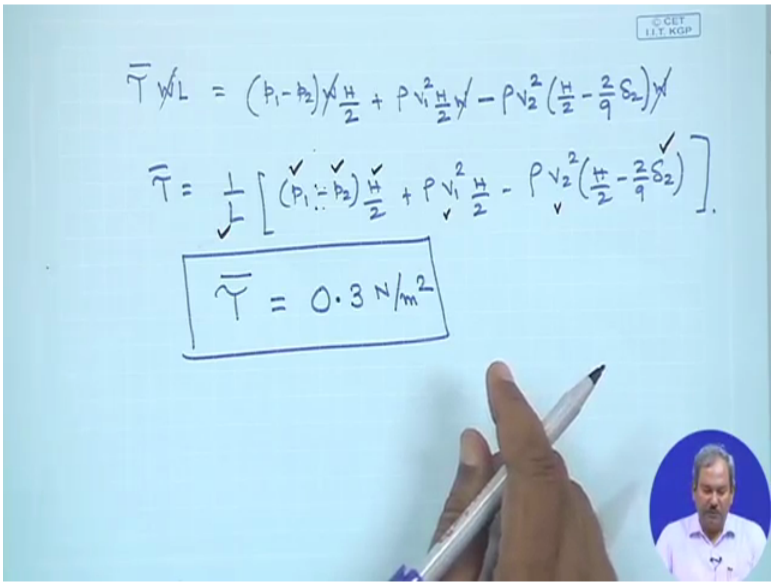


The 2nd part would be V2 and then rho V2 H by 2 - delta 2 times W. So if you look at this part, this entire thing inside the brackets gives you the mass flow rate out of the, this portion that is from 100 millimetre all the way to a 150 millimetre, from H by 2 - delta to, that is the length, that is the thickness through which the fluid, the air is moving with a constant velocity equal to V2. Also you note since the mass flow rate is going out of the control volume, both these terms are going to be positive.

Unlike the 1st part, since mass is coming in, it is going to be negative. So let us try to, let us try to evaluate this term 1st. So what we have then is 0 to delta 2 u rho u times W dY. And we realise that we have a relation which is u by U to be equals Y by delta to the power one 7th, that is the one 7th power laws which we are going to plug in here and this would result in rho V2 square delta 2W from 0 to 1 Eta to the power one, it to the power to by 7 because we have U square times d Eta, where Eta is simply equals Y by delta.

So this integration would simply change to once you put the, plug-in the expression of u in both cases, it would simply be this and this would be equal to rho V2 square 7 by 9 delta 2 times the 2. So I have evaluated this term, these 2 terms are straightforward which can now be calculated but our aim is to find what is the value of Tao bar, the average shear stress.

(Refer Slide Time: 22:05)



So we bring the average shear stress on one side and all the other terms on the other side and what you get is Tao bar, the average shear stress multiplied by the area on which it acts on as a function of the pressure, the surface force due to pressure from my previous expression + rho V1 square H by 2 times W which is the momentum that comes into the control volume through the control surface at location 1. And then the amount of momentum which goes out of the control volume as a result of flow through the boundary layer and flow through the remaining portion where the flow can be treated as inviscid.

So your Tao bar would simply be equal to1 by L , the W will cancel everywhere, from all terms, so 1 by L V1 - V2 multiplied by H by 2+ rho V1 square H by 2 - rho V2 square, H by 2 - 2 by 9 delta 2. In this you know what is the value of L, that has been given here, we already have evaluate what are P1 and P 2 you know the value of H, you have calculated V1, you have calculated V2, the value of delta 2 has also been provided in the problem.

So therefore the average shear stress is acting between location 1 and location 2 for a horizontal duct, you should be able to calculate the numerical value of the average shear stress to be equals 0.3 Newton per metre square. So this is a good problem, an ideal problem to demonstrate the utility of momentum integral equation even for solving the case of boundary, turbulent boundary layer. So this would really help in obtaining a quick solution because as engineers you would be mostly interested in the value of the shear stress or the force.

What is the force an object when it is immersed in a fluid and when there is a relative velocity between the solid and the liquid. So a simple balance of momentum would give you the value of the average shear stress. So I am sure if there are any questions are regarding this problem or anything that I have taught, I have taught you so far, you would be able to interact with me and if there are any doubts, I will clarify, or the teaching assistants of this course would be able to clarify any doubts that you may have.

What we have, what I am going to do next is something very interesting. It is commonly we understand what is drag, if you if you want to work or run on a windy day, you would feel different forces, you have to exert more if you are going against the wind or less if you are going with the wind. So these kinds of concepts play an important role in many applications, in the design of an automobile, in the design of a bus, in the design office spacecraft and various other forms of our daily lives.

When something moves let us say for example through air, what happens is the boundary layer is going to form on the solid object, on the blunt object and at some point there would be separation of the boundary layer and wakes are going to form. So wakes are formed when the boundary layer detaches from the surface and therefore the wakes are always going to be a low-pressure region. So if you if you move an object, a regularly shaped objects in air, there would be the formation of the wakes at the back of the moving object which would create a low-pressure region at the back of it.

So if you move it and if you have a low-pressure region here and a high-pressure region over here, then there would be a pressure force which will act against the flow. High-pressure, low-pressure, so the formation of the wakes, which are direct result of separation of boundary layers from the blunt object at a certain point, they will create an additional resistance to flow. And this kind of additional resistance is extremely important if you want to, if you want to do an efficient design of removing moving object in air.

There are beautiful examples of this from various fields of science, various fields of sports and so on. If you notice Formula One race, the car racing, what you would see is that there would be a lead car and other cars which are following the lead car. The lead car when it moves, it creates a wake at the back of it, the wake is a low-pressure region, so you would always see that the car which follows the 1^{st} car will always try to be, try to have its nose in the wake formed by the 1^{st} car .

So what happens is then the 2^{nd} car would experience less of a pressure drag because its nose is exposed to a region of low-pressure. And the car which is following the 2^{nd} car will also try to be in the wake formed by the 2^{nd} car and so on. So this will continue for quite some time and the wear and tear on the tyres of the 2^{nd} car would be comparatively less as that of the 1^{st} car. So the pack moves on like that and the 2^{nd} car or the 3^{rd} car or the car behind would try to overtake only at the last possible moment.

So for a very long period the cars would follow each other, only when the finish line is in sight, it would try to overtake and move to the front. And since it has conserved its energy, it has conserved, it is his tyres and everything are in relatively better shape, better condition than the 1^{st} car, if it can overtake the 1^{st} car, then it will win the race. The same thing you would observe when you look at the, look at cycling. So whenever there is a race, cycle race, there would always be cycles, bikes, which would try to be in the wake formed by the cyclists just in front of it.

So by judicious application of your fluid mechanics and your concept of boundary layer, you would be able to win the race. So that is very interesting. So this part of the course, this part of the class I am going to talk about drags, the drags are of 2 types, one is frictional drag, the 2^{nd} is pressure drag. So whenever an object moves in air, it has a friction drag because of its interaction with the air above it due to viscosity and the pressure track which is a function of the shape of the object.

So if you look at the shape of the bullet train, the nose of the bullet train, the engine of the bullet train, it is designed in such a way to reduce the pressure track. It is designed in such a way so that the boundary layer separation is delayed and therefore the formation of a low-pressure wake or region at the back of the train or at the back of the engine is minimised, such that the opposite pressure force will can also be reduced.

So from the design of the nose of a rocket, nose of a bullet engine, engine of a fast moving train, car racing, cycling, and in so many other ways, boundary layers wakes and drag, they form an integral part in the aerodynamic design of all these objects. So it is very important that we have some idea of what is drag and we would introduce the concept is similar to friction coefficient what is the drag coefficient.

Because everywhere you would see that the results experimental results are reported in the form of drag coefficient and obviously something any object which can be modified geometrically or otherwise to result in the lower drag coefficient would be the preferred design. So any outcome of the design would probably be manifested, represented by a reduction in the drag coefficient.

So we need to know what is that coefficient, we will restrict ourselves in this course to the friction drag only, so what is the drag coefficient, what are the expressions of drag coefficient in laminar flow as well as in turbulent flow, what is flow rate, flow separation and is there a way to use drag, the laminar flow drag, turbulent flow drag, combination of these 2, is there a way to use them to achieve certain things and I will give an example from sports. At the beginning he of this course I said that when a fast bowler bowls a swing ball where the ball will change its trajectory in air, how does he do that?

So we would try to give you a partial answer of that based on our concept of drag, drag coefficient, laminar flow, turbulent flow, separation of boundary layers and so on, so that I would introduce in the next class.