

Introduction to Boundary Layers
Dr. Rinku Mukherjee
Department of Applied Mechanics
Indian Institute of Technology, Madras

Module – 04
Lecture – 08
Control Volume Approach to Derive
Expressions For δ and Θ over a flat plate

Hi. Welcome back, so let us finish this problem that we started talking about in the previous module. So, we will go to design or look at designing a wind tunnel.

(Refer Slide Time: 00:31)

The image shows a digital whiteboard with handwritten notes. The notes are organized into three sections: Assumptions, Properties, and Analysis.

Assumptions:

- ① Flow is steady + incompressible
- ② The walls are smooth.
- ③ Disturbances + vibrations are kept at a minimum.
- ④ BL is laminar.

Properties: $\nu = 1.507 \times 10^{-5} \text{ m}^2/\text{s}$

Analysis: $Re_x = \frac{Vx}{\nu} = \frac{40 \times 0.3}{1.507 \times 10^{-5}} = 7.94 \times 10^4$ or 0.0794×10^6

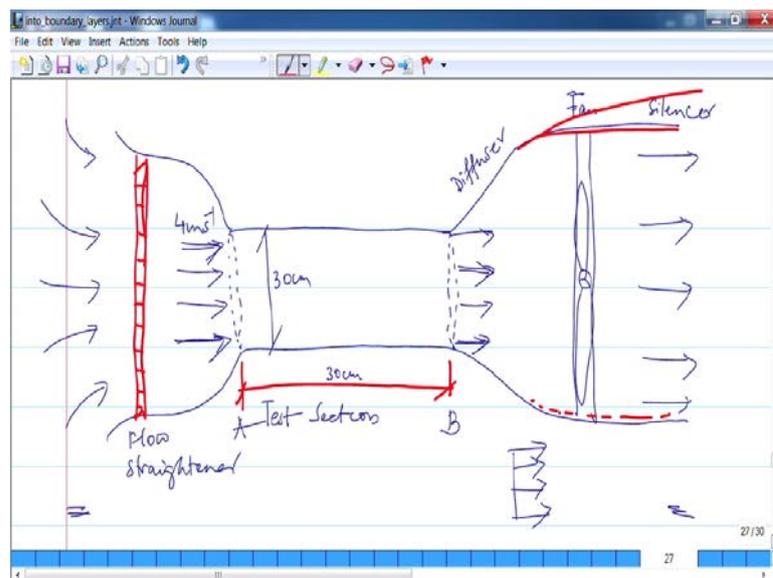
Flow is laminar thro-out the test section.

Now, let's start like this and now, we always start out while making assumption. We do start out by making assumptions. Now, what are these assumptions? Number one, the flow is steady and incompressible. What is that mean? That throughout when the flow is happening throughout the wind tunnel, we can take a single value of the density. There are no drastic changes in the density as the fluid is moving into the wind tunnel, into the test section, out of the test section, through the fan, through the silencer and out; there is no change in that. And the flow is steady, so if I would look at the flow at sudden instance of time I would not be seen drastic difference in its properties.

The walls are smooth. Well, the walls are smooth so that we are not going to look at any effects of the wall in this case. What we would be concern about, is primarily the fluid. I mean moves of the walls and what kind of behavior that makes. So we are not taking into account any property of the walls. Although, practically that is quite not possible, the walls will be angulated, walls will be rough while walls will have small partitions here and there. And there might be walls due to it might not be even clean. So you have to keep it clean as much as possible. But those are things which could play into behavior of your fluid. So, we are not going to that I mean we are just going to see you that the walls are smooth.

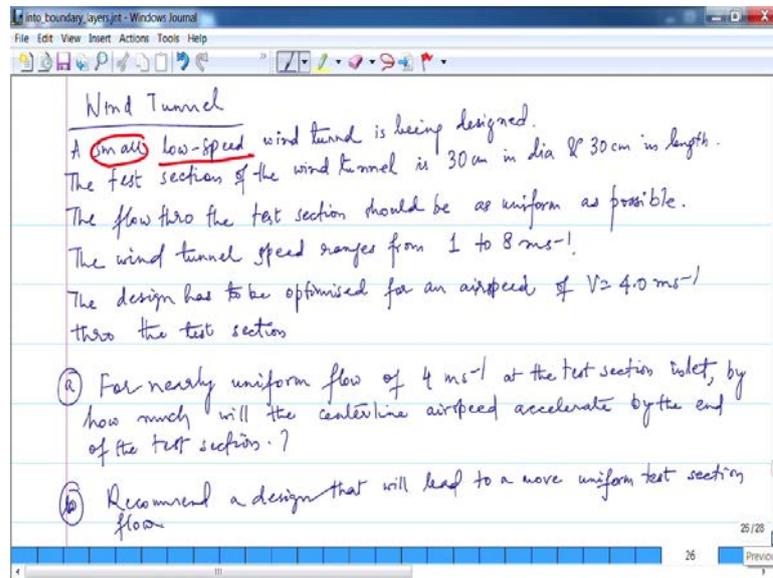
Disturbances and vibrations are kept at minimum correct. So, disturbances and vibrations are kept at a minimum and hence given whatever that the boundary layer is laminar. These are basically my assumptions and the properties based on what is given to us, what we basically get is that μ is 1.507×10^{-5} meter square per second, so this ν . Now, let us go head and try to solve this. First things first, let us calculate Re_x and that is; which is equal to, in this particular case what is V , V is the incoming velocity which is given to us to be $4 \times X$.

(Refer Slide Time: 04:58)



What is X ? X is distance which is traveled and what is given to us.

(Refer Slide Time: 05:04)

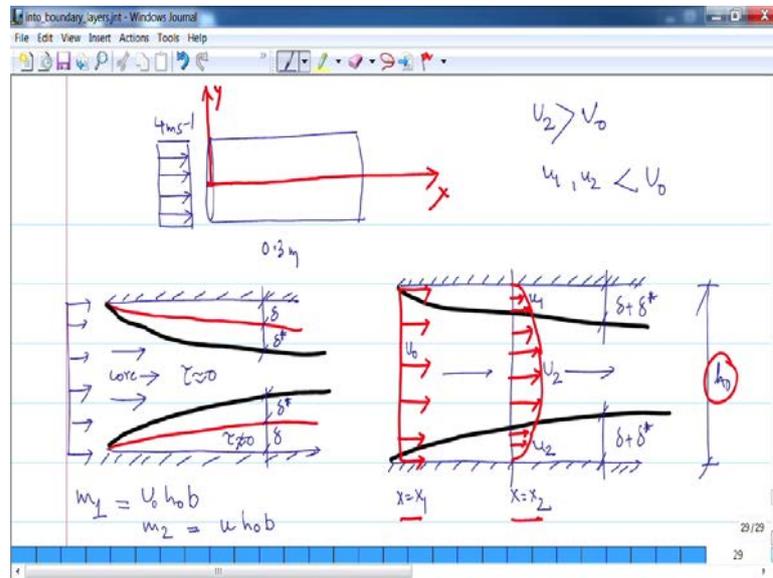


What is given to us is that the test section of the wind tunnel is 30 centimeter diameter and 30 centimeters in length. So, basically what we saying that this is 30 centimeter and this is 30 centimeter this length, I have a flow which is moving from here through this traversing the whole lines and coming out of this, so this is the entrance of the test section lets call this is A and B section. Section A is the entrance, B is the exit of the test section so that I am trying to find out the reynolds number at the end of the test section, so that X basically means it is 0.3 meters and mu is given above, so 1.507×10^6 to the power minus 5, what I get is 7.94×10^4 . Usually, I would write this as say this is 0.0794×10^6 . It is about 0.08 million and we are less than 5 million. So this is a laminar boundary layer. Therefore, the reynolds number at the end of the test section is around is laminar, which means that the flow is laminar throughout the test section.

Now there is something very interesting here, before I do the math, so I am going to do is discussion little bit. What is going on here? So now the question is that, is it simple enough that I will have flow which comes in here then it will go out here, what is the problem? You know why do we have to? And then there is this also question here, that for nearly uniform flow 4 meter per second at the test section inlet, by how much, will

the centerline airspeed accelerate by the end of the test section? What is that even mean? So, what is asking is if this is 4 meters per second and nearly uniform, so nearly uniform.

(Refer Slide Time: 08:02)



Basically now, if instead of break this down, we say I have a pipe like that. This is essentially my test section and this is a 30 centimeters, so this is 0.3 meters and this dia is 0.03 meters, and what I have here is nearly. So I have uniform flow which is coming in and this is 4 meter per second. Then the question is how, will the centerline, say I am going call this as x say y. So the question is how will the centerline velocity accelerate? What is that even mean? I mean why we are saying that, this is going to accelerate. Let us look at this; before we start even doing this, let us look at this. Now, you would have the same case, so if you have flow I am going to exaggerate this a little bit.

These are basically walls. These are the walls of the in this case have test section or pipe or whatever it is, and you have a free stream which comes in. You have a uniform free stream which comes in. So, in the normal case what will happen is, you will have free stream which will just continue to go, but however, that is not, what we does not seem to be happening. Now what is the case here? Now what is happening here? Is that what we would have is that there will be a development of a boundary layer on both sides and lets me call that, so say here at some location, now this height is basically nothing but delta.

Similarly, this height is nothing but δ . Now, because of the existence of this boundary layer, a streamline like we have discussed in the in the couple of modules back, so this streamline when it comes here, it is going to be pushed away from the wall because of the existence of the boundary layer.

Similarly, at the lower wall, if it comes here, it is going to shift it away from itself from the lower wall, because of the existence in boundary layer. Therefore, I have a streamline from say at this particular point, so I could actually draw another thing like that and I could draw another sort of thing like that. Such that, that if I would move at any location, for example at this location, what will happen is at this location again, so this is the part and this is nothing but the displacement of the thickness. This is the distance by which the streamline is going to be displaced away from the boundary layer. Now, this is a flow picture, where this is actually supposed to be the core area. This is actually called the Core, the flow is moving like this. So, you have enough flow which is basically moving like this and this is the region where shear stress is not 0 in existent, and this is shear stress is 0. This here is the inviscid or irritation of flow, but this is the boundary layer, so viscid effect terminating in this region.

Now, let us understand this just little bit more. Again, if I where to draw this. So now, here I have this, this is my wall and here are the two walls of my test section here. Let me also then do this. So then, what is this distance all about? This is nothing but δ plus δ^* . Similarly, this is δ plus δ^* . Now, what I am going to do now is basically, in our case this is the diameter, so let us say that this height. This height is say, h_{naught} . This height is nothing but which is h_{naught} .

Now, if I were to draw a velocity profile right here at the entrance, say at this point. Now if I do this, let us draw a velocity profile right at the entrance, somewhere here. If I do this, I am going to draw that, so this is my velocity profile at this location. Now, this location say X is equal to say X_1 .

Now, let us come to another location which is say X is equal to X_2 . Now the question is, how will the velocity profile look like? And I thing I had ask and posed a question in couple of modules back is that say this velocity here, let the value of this be see U

naught. What happens if I move along X? If I move along X like that, then what happens to this U_{naught} ? Does it increase or does it decrease. Now if you look here, the total mass flow at the section X is equals to X_1 is what? So let me call that say, m_1 and that is going to be say, $U_{naught} h_{naught} b$, which is just the breadth. This is the mass flow. This is a total mass flow to X_1 .

Now, we clearly know that the velocity here through this is going to be something like this. And the velocity from here to here is also going to be something like this because of the existence of the boundary layer. That means, say this is the δ here, so the corresponding height here we are not going to be able to keep the velocity at U_{naught} . This velocity is going to be less than U_{naught} is not it. For example, here it is U_{naught} at the wall right here, but it is 0 here at X_2 . Similarly, here it is U_{naught} , but this thing is less than U_{naught} . However, if I have to apply mass conservation at these two sections, the total mass flowing at section X_1 through this height h_{naught} should be the same at X_1 as well as X_2 . Which means that m_2 should be equal to velocity $h_{naught} b$.

Now, if that as to happen we know that in near the walls the velocity is less than U_{naught} and it is only become U_{naught} away from the walls at distance $\delta + \delta^*$ from whatever I can see here. In that case therefore, this velocity as to increase. This velocity has to be more than U_{naught} to compensate for the whole thing. So that when I have this therefore, my mass is conserved, which means that here the velocity is say U_2 . So basically, what I am saying is that U_2 is larger than U_{naught} . And within this within the displaced bound within the boundary layer thickness and displaced thickness here. So here basically, if I say take U_1 and if I take this as U_2 , so both U_1 and U_2 is less than U_{naught} . I think that make sense.

So therefore, this question that how does the centerline velocity accelerates? What is the acceleration? So the first question is, why will it accelerate in the first place? So this is the reason. Therefore, what is happening is you can also look at here, another way of looking at this displacement thickness that it is all most like an extra of thickness of the wall itself. So that is almost like behaving like another layer, a thick wall. The path, the free path available for the flow to move is decreased, but it as to keep the same mass

which is moving through the two sections. Therefore, it will lead to accelerate, so that low of mass consideration is valid. Hence, in the core region flow is basically accelerating. So that is what we mean.

(Refer Slide Time: 21:02)

The image shows a digital whiteboard with handwritten notes and diagrams. At the top, there are two equations for displacement thickness δ^+ :

$$\frac{\delta^+}{x} = \frac{1.72}{\sqrt{Re}}$$

$$\delta^+ = \frac{1.72 \times 0.30}{\sqrt{7.94 \times 10^4}} = 1.83 \text{ mm}$$

Below these are two diagrams of a circular pipe. The left diagram, labeled 'Entrance to the test section' and 'A', shows a circle with radius R . The right diagram, labeled 'End of the Test-section' and 'B', shows a circle with an inner radius $R - \delta^+$ and an outer radius R . To the right of the diagrams are equations for continuity and velocity:

$$V_B \text{ area}_B = V_A \text{ area}_A$$

$$V_B = V_A \frac{\text{area}_A}{\text{area}_B} = 4.0 \frac{\pi R^2}{\pi (R - \delta^+)^2}$$

$$V_B = 4.1 \text{ ms}^{-1}$$

The whiteboard interface includes a menu bar (File, Edit, View, Insert, Actions, Tools, Help), a toolbar with drawing tools, and a status bar at the bottom showing '30/30' and '30'.

Now, going back for the problem; then delta star, this we done earlier. At the end of the boundary layer, here basically you can take delta star is 1.72 and x is the end of the test section and this is something that we have calculated, which is 7.94 10 to the power 4. What I get is, 1.83 millimeters. Now, what will do is look at this here, when we are here. So, what will do is we will look at the cross sectional view of both these the inlet as well as the entrance as well as the end of the test section. Why? If I where to look at this for example.

For example, let me just go back here. This is nothing but the radius and this is given to be 15 centimeters and the diameter is 30 centimeters. So this is the entrance to the test section. And now what happens, this is end of the test section. Now like I said, this boundary layer displacement thickness here, now this actually behaves like as if there is an extra thickness to the wall, so that it will be displaced from there. So the end of the test section therefore, if I am where to exaggerate that if the flow is going to be displaced

from the wall, by how much? By delta, so this distance is nothing but delta star. This is nothing but R minus delta star. So this is nothing but delta star.

However, right from the entrance to the end the conservation of mass has to be valid. Therefore, I can write the very simple equation. This is section A and this is section B, so now, V_B area B. This is basically velocity and is same fluid so row is same, so you just at a multiple row so I am canceling that out is equal to V_A area A. So, then basically we need to calculate V_B , which is equal to V_A area A by area B. Now, V_A is 4 which is given to us. Area A, What is area A? It is πR^2 and area B is $\pi (R - \delta)^2$. So, using this, what I get is V_B is essentially 4.1 meter per second. So delta star is this value, I get that. So, essentially what I see here is that the air speed increases approximately by 2.5 percent. The air speed increases by 2.1 percent so the test section due to the effect of the delta star. So it does increase by that.

Now, the other question was that, what would be a good way to design the wind tunnel so that we get more or less uniform test section flow. Now, this is basically just for your knowledge. The kind of things one would do is sorry, let us go back here. So, what we understand here? In this particular case the way we have here, we have a diffuser and then this wall straightens out. But what we see is that, because of the boundary layer now the displacement thickness it almost acting like a thickness of the wall. So in that case, because of that it is going to not give you this entire area for the flow to move. So instead of keeping this horizontal like that so you could diverge it, so that I get little more space for the flow to move. So that is one of the things one can do.

Another thing is which you can use a suction pump and suck out all this air which is sitting on the walls here and creating a boundary layer and which is displacing the air which comes here into a here, which comes near the walls away from it. So that is also a possibility. Now, that is something is also expensive so one will have to be careful. I think will stop here and I wanted to do another example, because I thought that will be interesting but hopefully we can another time. So, for now I think we will stop and take this up again in the next module.

Thank you.