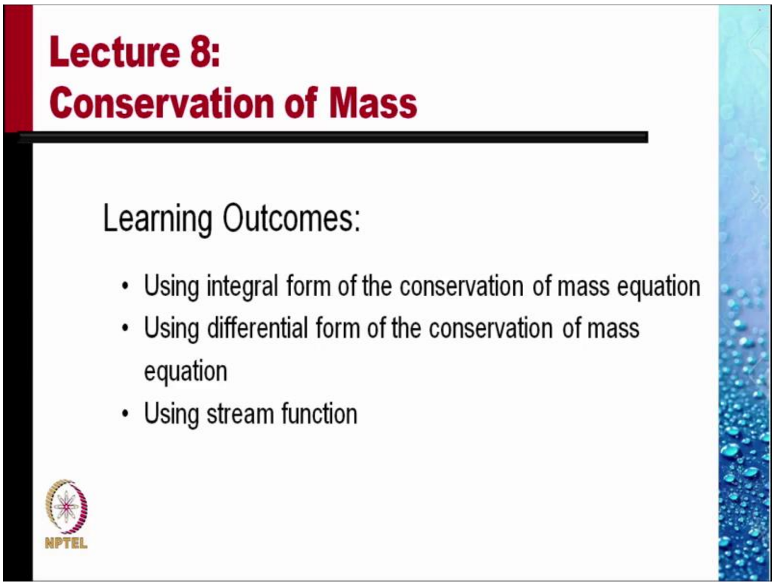
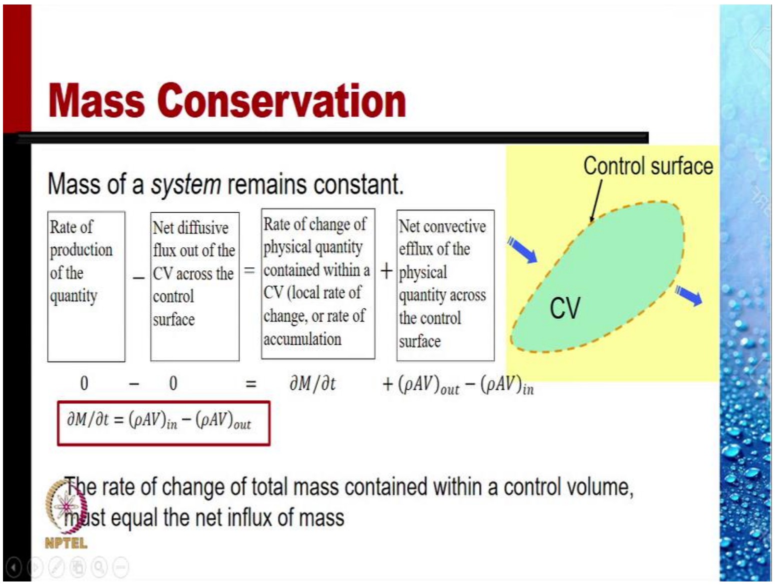
Fluid Mechanics and Its Applications
Professor. Vijay Gupta
Indian Institute of Technology, Delhi
Lecture 8
Conservation of Mass

(Refer Slide Time: 00:26)



Welcome back. In today's lecture, we will discuss conservation of mass equation.

(Refer Slide Time: 00:30)



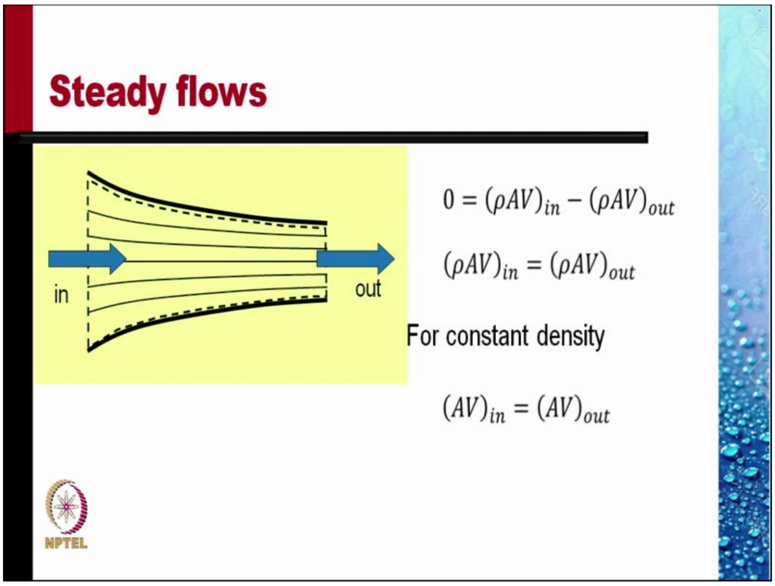
The basic principle of mass conservation states that the mass of a system remains constant. This is a law that is available for a system, that is, for fixed material particles. As you know, that we like to use control volumes in fluids. So, we will have to find a conversion of this law to control volumes. And for this, we use the Reynolds's transport theorem. We have a control volume CV with the control surface. The mass can enter and cross the control surface.

As we discussed in the last lecture, the Reynolds's theorem says that the rate of production of a quantity, a physical quantity that we are dealing with, minus the net diffusive flux out of the control volume across the control surface should equal the rate of change of physical quantity contained within the control volume, that is, the local rate of change, or the rate of accumulation, plus the convective flux of the physical quantity across the control surface.

When the property is mass, the rate of production of that quantity is obviously zero, if we rule out nuclear reactions. The diffusive flux in the case of mass is also zero. And the rate of change of physical quantity contained within the control volume, which is the local rate of accumulation is $\partial M/\partial t$, and the last, the net convective flux of the physical quantity across the control surface is $(\rho AV)_{out}$ minus $(\rho AV)_{in}$, the net flux.

This can be manipulated to give d $\partial M/\partial t$, the rate of accumulation of mass within the control volume is equal to the net influx of mass across the control surface. This is the form of mass conservation equation that we use in fluid mechanics.

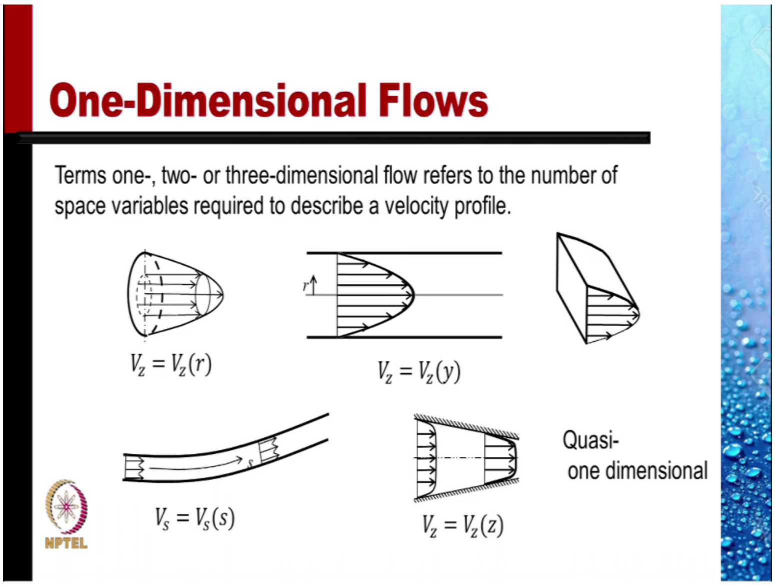
(Refer Slide Time: 03:14)



Let us consider a steady flow in a channel where the mass is coming in at the left end and is leaving out at the right end. Clearly, in steady flows, there will be no accumulation of mass within the control volume. So, the net influx written here is zero $(\rho AV)_{in} - (\rho AV)_{out} = 0$.

For constant density, when the density of fluid does not change, ρ_{in} is the same as ρ_{out}. So, $(AV)_{in}$ is equal to $(AV)_{out}$: A_{in} the area of the channel at the inlet, $V_{in}$the velocity at the inlet, and similarly, A_{out} is the area of the channel at the outlet and V_{0ut} is the velocity at the outlet.

(Refer Slide Time: 04:27)

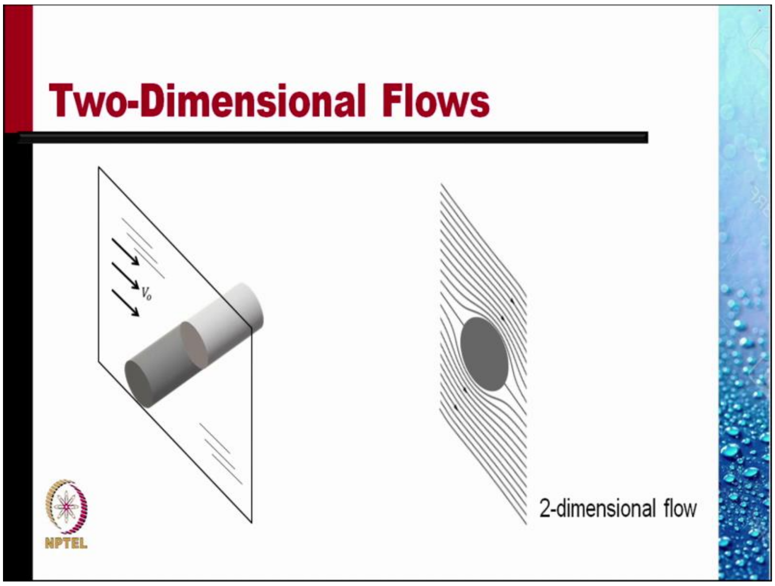


The terms one, two or three-dimensional flows, refers to the number of space variables required to describe the velocity profile. This conical shape represents the velocity profile in a pipe, where V_z, the velocity along the *z*-axis, is a function of *r* alone. We had discussed earlier that if the flow is laminar, this profile is parabolic. It varies like r^2. We need only one variable *r* to describe this profile. That is why this is called a one-dimensional profile.

Similarly, if you have a channel made of two parallel plates, the velocity profile V_z is a function only of *y*, the distance from the central axis. This is also a one-dimensional profile. If the channel is not straight, but curved, but it is thin in the sense that the cross-sectional dimension is small compared to the dimension that we measure along the axis, then we can write the velocity along the axis as a function only of *s*. This again can be considered to be a one-dimensional profile.

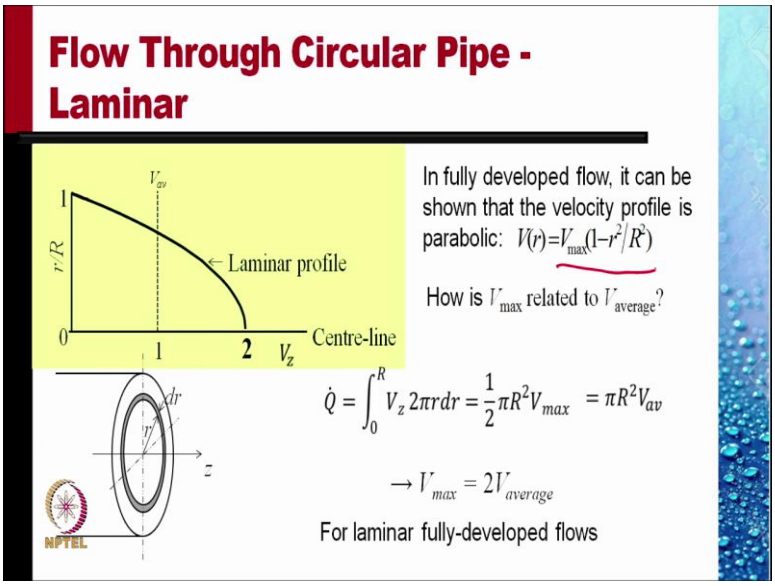
If we have a slowly tapering channel, the velocity is no longer a function of only one variable because the velocity near the walls also changes in the *y* direction. So this is not truly a one dimensional flow, but if the boundary layer where the velocity varies is thin, then the flow can be treated as a quasi-one dimensional flow.

(Refer Slide Time: 06:53)



Consider the flow past a circular cylinder with a velocity V_o. If we take a plane perpendicular to the axis of the cylinder and look at the flow in that plane, we will see the flow within the plane to be like this. This clearly is a function of two space variables, a vertical variable and a variable along the flow direction. This flow pattern is independent of the third dimension, that is, we could take the plane anywhere along the axis of the cylinder and we will observe the same flow. So that is why velocity profile does not depend upon the third dimension, it is called a two-dimensional flow.

(Refer Slide Time: 08:01)



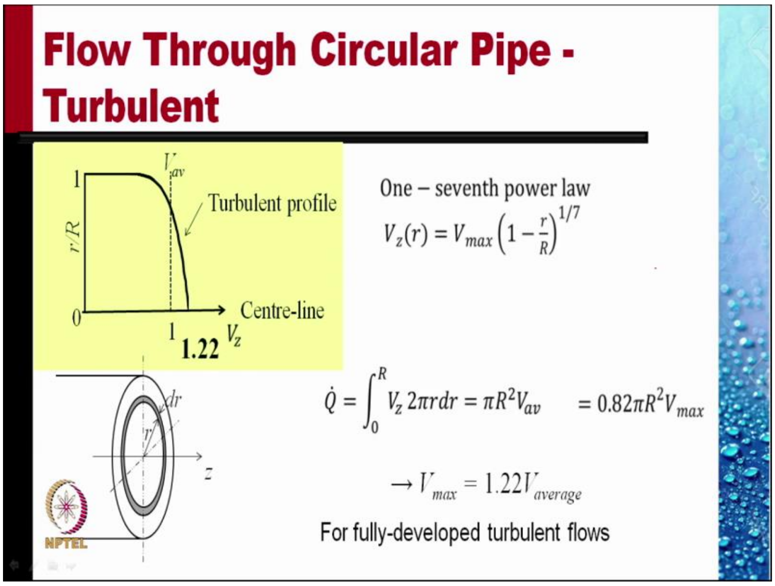
In a circular pipe and if the flow is laminar, the velocity profile is parabolic and is given as $V = V_{max}\left(1 - \frac{r^2}{R^2}\right)$, where r is the distance measured from the axis in the radial direction, and

R is the radius of the tube, V_{max} is the velocity at the centerline of the tube, where it is maximum.

How is V_{max} related to $V_{average}$? To calculate $V_{average}$, we need to find out what is the flow that is taking place across the whole section of the tube, and divide that by the area of the tube. And that will give you the average velocity. The process is that we consider a ring of thickness dr at radius r. The velocity there is given by the formula given here, $V = V_{max}\left(1 - \frac{r^2}{R^2}\right)$.

And so, we can set up integration. The volume flow rate is the integral over r from 0 to capital R. V_z is the velocity at that location, which is $V = V_{max}\left(1 - \frac{r^2}{R^2}\right)$, into the area of this ring $2\pi r dr$. In this, where the integral comes out to be $\frac{1}{2}\pi R^2 V_{max}$, after putting in the value of V_z given by the velocity profile and integrating. This is equated to $V_{average}$ times the area of the pipe, πR^2. And so, clearly we get $V_{max}$is equal to twice the $V_{average}$. The average velocity is one half of V_{max}.

(Refer Slide Time: 10:40)

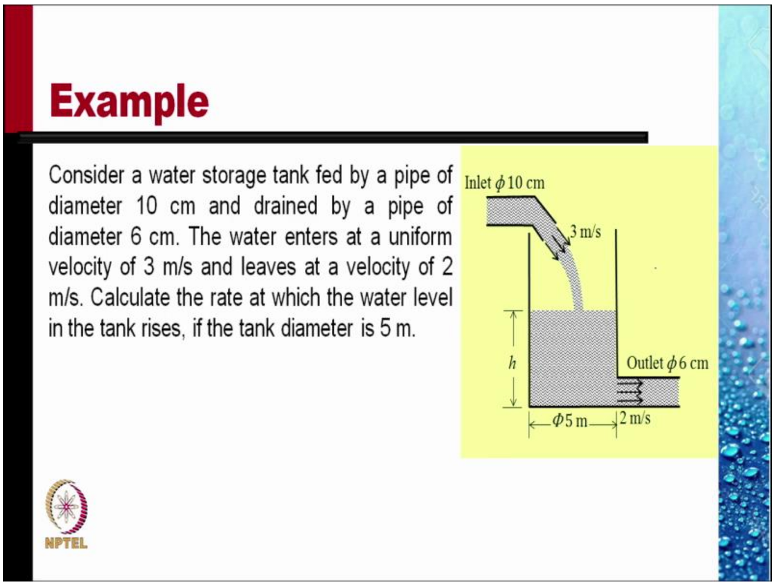


Let us again do this problem with a turbulent velocity profile. One of the most commonly used turbulent velocity profile in a circular tube is the one-seventh power law, V_z at any radius r is $V_z = V_{max}\left(1 - \frac{r}{R}\right)^{1/7}$, and we find $V_{average}$ in the same fashion.

We take a ring of thickness dr at the radius r. We write the volume for it $\dot{Q}$ as integral over 0 to R of V_z times the area of the ring, which is $2\pi r dr$, and this is equated to $V_{average}$ times πR^2, and from this we get $\dot{Q}$ as $0.82\pi R^2 V_{max}$, and then we get $V_{max}$is equal to 1.22 $V_{average}$.

So, V_{max} in the laminar law is twice the $V_{average}$, but in turbulent flow it is 1.22 times $V_{average}$. That is because the velocity profile is much flatter than the parabolic velocity profile of laminar flows and extends very close to the walls of the tube.

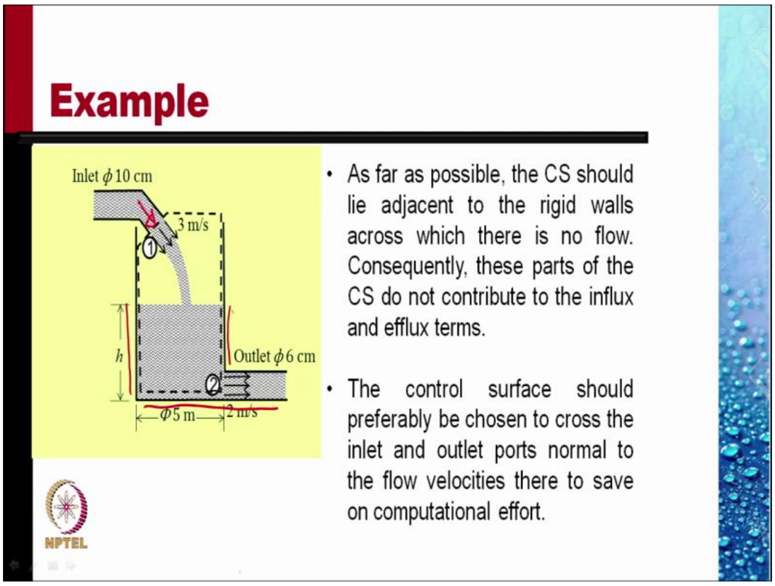
(Refer Slide Time: 12:22)



Let us do an example of the use of the mass conservation equation that we obtained for a control volume in this. Consider a water storage tank fed by a pipe of diameter 10 cm is drained by a pipe of diameter 6 cm. The water enters at a uniform velocity of 3 m/s and leaves at a velocity of 2 m/s. Calculate the rate at which the water level in the tank rises, if the tank diameter is 5 m.

This is not a steady flow example. So we take an appropriate control volume. The rate of accumulation of water in the control volume is equal to the net rate of influx of water into the control volume across the control surfaces. So we have to choose a control volume that is appropriate.

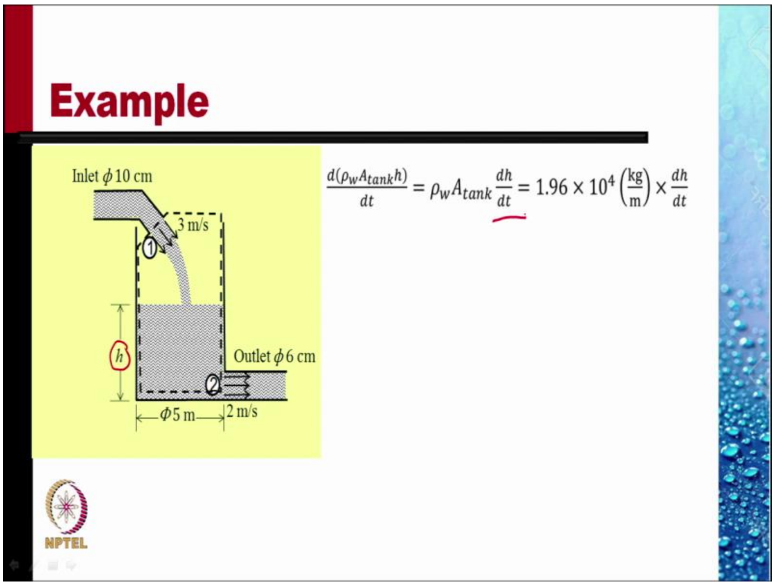
(Refer Slide Time: 13:37)



As far as possible, the control surface of the control volume should lie adjacent to the rigid walls across which there is no flow. Like the portions here, here and here. Why? Because they are adjacent to rigid walls, and so, there is no flow across these parts of the control surface.

The control surface should preferably chosen to cross the inlet and the outlet ports normal to the flow velocity to save on computational effort. Because if the surface normal, the flow rate is simply velocity times the area of the flow. So the appropriate control volume is something like this, shown by broken lines. This portion at 1, the control surface is perpendicular to the velocity of water. In this portion at 2 is also perpendicular to the velocity of water.

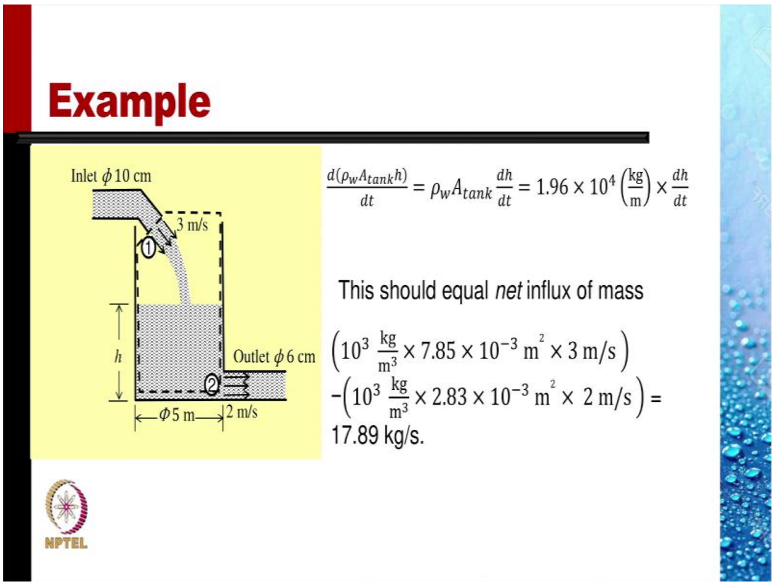
(Refer Slide Time: 15:15)



So, if we apply the mass balance equation, the first term is rate of accumulation of mass, which is the rate at which mass of water contained within the control volume changes. The mass of water within the control volume is ρ_w, the density of water, times the cross-sectional area A of the tank, and the height h up to which the water is stored in the tank. This is expanded as $\rho_w A_{tank} \frac{dh}{dt}$. Then plug in the value ρ_w and A_{tank}, I get 1.96×10^4 kg/m into $\frac{dh}{dt}$. This is the rate of accumulation of water into the tank.

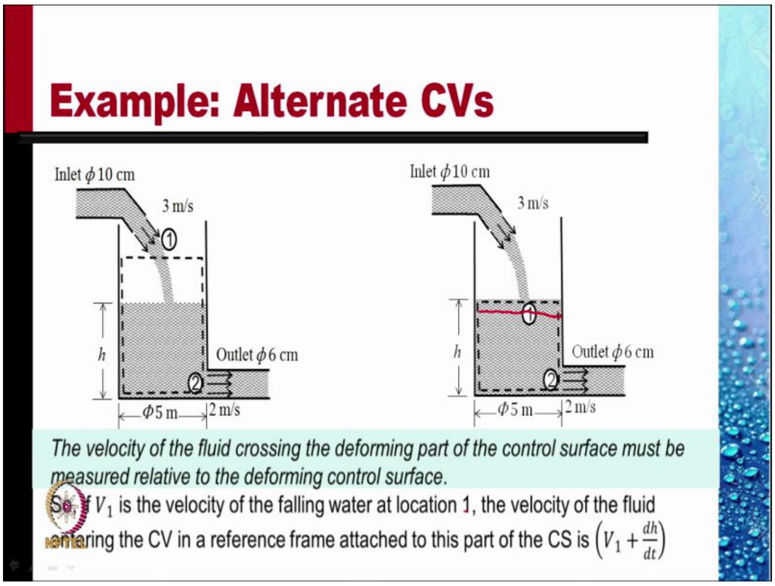
This should be equal to the net influx of water into the tank across the control surfaces. There is influx across 1, and there is a efflux across the surface marked 2. So, the naet flux is the water crossing the surface at 1 minus the water crossing the surface at 2.

(Refer Slide Time: 16:53)



At 1, the density into the area (10 cm diameter pipe) translates to 7.85×10^{-3} m^2 into velocity, which is 3 m/s, minus the efflux at 2, which is density times the area (6 cm diameter) times the velocity which is 2 m/s there. So, the net influx or mass is 17.89 kg/s. This is equated to the rate of accumulation to obtain $\frac{dh}{dt}$.

(Refer Slide Time: 17:45)



We could work with alternate control volumes as well. Suppose we have chosen this control volume as shown. We are not worried about the second recommendation which said that the control surface should preferably be normal to the velocities. At 1, the velocity is inclined at 45° to this. This would complicate the calculations. The area would now increase to the area of the pipe divided by cos 45°.

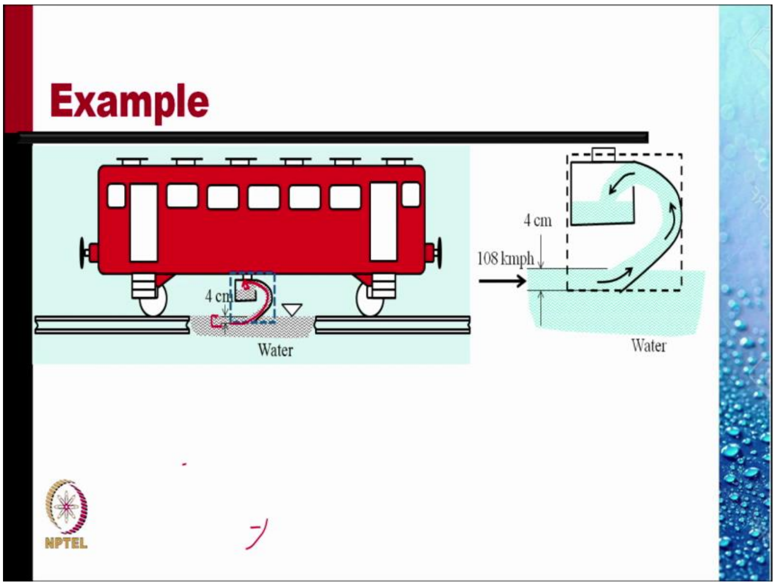
But, when we find the flow we will use only the velocity, the vertical component of velocity, would contribute to flow across that area, and the vertical component of velocity would be V times cosine 45°. So, the product of the velocity and area would still be the same VA. So, we should get the same result.

Another example of a control volume is a little bit more tricky. Here we take a control volume where the top surface is taken to coincide with the top surface of the water inside the tank. As more and more water accumulate in this tank, this part of the control surface would move. The control volume would expand.

Can we apply the mass equation to this control volume? Yes, we can apply, but we must realize that while calculating the influx and efflux, we must take the velocity of the fluid relative to the control surface. Here the control surface is moving up at the rate dh/dt. And so, the average velocity of influx at that surface would be velocity of water plus dh/dt, the velocity at which the water is crossing into the control volume, the relative velocity. To state again, the velocity of the fluid crossing the deforming part of the control surface must be measured relative to the deforming control surface.

So, V_1 is the velocity of the falling water at location one, the velocity of the fluid entering the control volume in a reference frame attached to this part of the control surfaces $V_1 + \frac{dh}{dt}$. And we use this equation we would again get back the same result.

(Refer Slide Time: 21:22)



Let us take another example. In very old days, the long distance trains used to pick up water for the use from a ditch, which was running parallel to the rails the water was scooped up as the train moved to the left, the water was scooped up into the reservoir. In this example, we have shown that the scoop is picking up a 4 cm layer of water.

What is the rate of accumulation of water inside the control volume? We choose a control volume like this. In this control volume, there is influx only at one point, only at one location, this location. If we take unit depth of this scoop, then the area of the inlet is 4 cm into a unit depth, 1 m. This is the area of this scoop and this area is normal to the velocity of water moving in.

The relative velocity of water across the surface is the same as the velocity at which the train is moving forward. So, we know the velocity, we know the influx of water, the density of water times the velocity of the train times this area 4 cm into 1 m per m depth. There is no efflux from this control surface, the rate of accumulation must be just this, the density of water and the velocity times the area of scooping.

This is the control volume that we use. The speed or the 108 kmph. There is no exit of water. So the water is continually being accumulated inside the tank, inside the control volume.