And once I know this area and once I apply these things to the momentum flux correction factor equations, I will have a this far.

$$\beta = \frac{1}{A_c}\int_A \left(\frac{V}{V_{avg}}\right)^2 dA_c$$

$$\beta = \frac{1}{\pi R^2}\int_0^R \left(1 - \frac{r^2}{R^2}\right)^2 2\pi r\, dr$$

$$given$$

$$V = V_{avg}\left(1 - \frac{r^2}{R^2}\right)$$

To do these integrations, we can consider

$$y = 1 - \frac{r^2}{R^2}$$

$$dy = -2\mathrm{r}\,\frac{dr}{R^2}$$

and in terms of y we are writing it to just do the integrations, nothing else. In that, if you look it, you have

y = 1 @ r = 0

y = 0 @ r = R

So we will change this upper limit and lower limit of the equations when we are converting from dr to the y integrations. So, this is what, 0 to 1, the -1 components are there, -y square will be there.

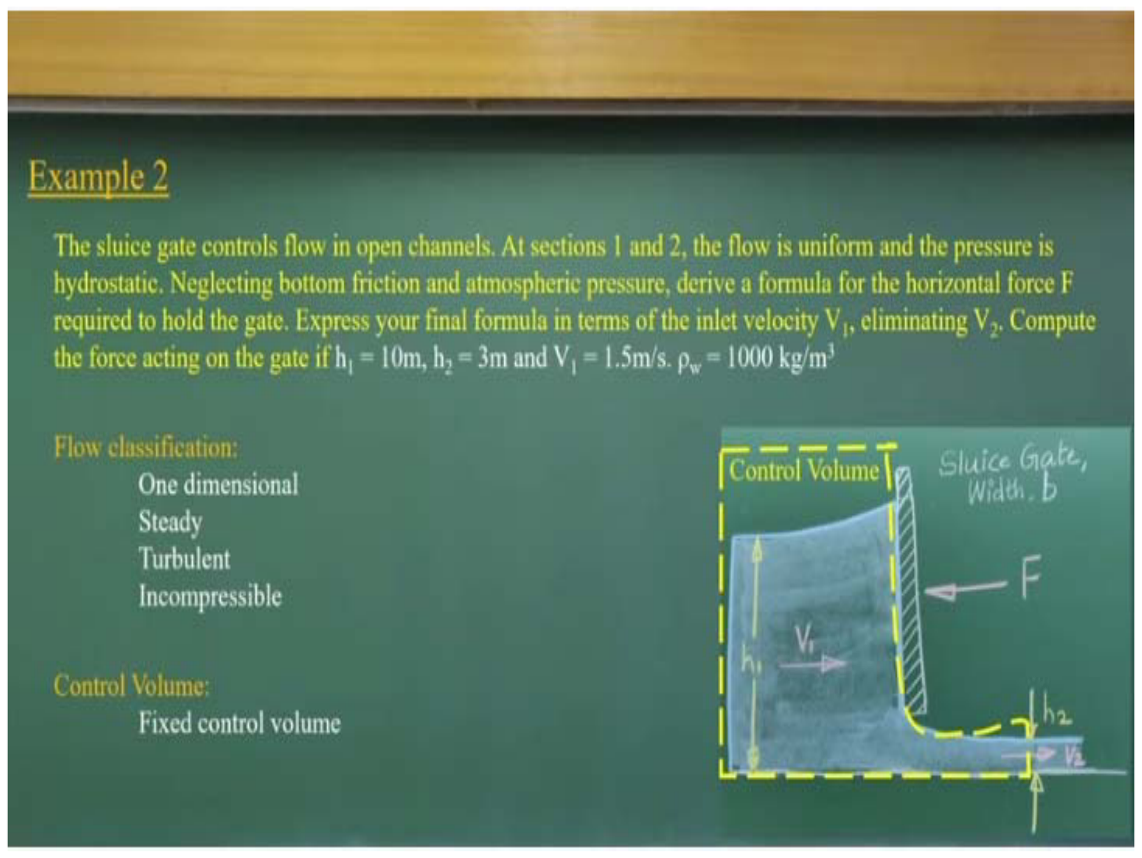
And if you substitute these values, and you will get it one by third. So, in laminar flow case whatever this velocity distributions, the beta factor is called, comes to what one by third. That means, if you computing the momentum flux using these average velocities, the actual momentum flux going through that surface if follow these velocity distributions, will be the one third of that. If you look it that, if you have beta = 1/3.

$$\beta = -\int_1^0 (y)^2\, dy \;=\; \frac{1}{3}$$

What it indicates that the momentum flux using velocity distributions divide by the momentum flux using average velocity. So, what it indicates that, the momentum flux using the velocity distribution will be the one third of the momentum flux using average velocity. The momentum velocity using the average velocity is much, much larger and that what is to be divide by one third to compute it the momentum flux using the velocity distribution.

So, you can know it, what is the importance of the momentum flux correction factors when the velocity distribution is not uniform. But in some of the cases, velocity distributions like for example for turbulent flow, this value is close to 1.01 or 1.04, so for the turbulent flow. So, in that case you may assume it, beta equal to the one, but in case of the laminar flow and all with you have the momentum flux correction factors are different, it depends upon the velocity distributions. What type of velocity distributions you have.

(Refer Slide Time: 40:53)



Now, let us come to the second example, which is very interesting example, which is almost all the Fluid Mechanics book have these examples with some numerical values are the difference. The problem is very interesting problems is that, there is a gate and the flow is coming from this side and going out through the gate here, the velocities V1 and V2 and h1 and h2 is the flow depth, this is the sluice gate.

[The sluice gate controls flow in open channels. At sections 1 and 2, the flow is uniform and the pressure is hydrostatic. Neglecting bottom friction and atmospheric pressure, derive a formula for the horizontal force F required to hold the gate. Express your final formula in terms of the inlet velocity V_1, eliminating V_2. Compute the force acting on the gate if h_1 = 10m, h_2 = 3m and V_1 = 1.5m/s. ρ_w = 1000 kg/m^3]

What could be the force, the horizontal force acting on this gate? So, that much of force necessary to hold it, okay. What will be the final formula, in terms of V1, V2, or eliminating the V2, if there is, this numerical value is there, that off stream flow depth is 10 meters, h2 is 3 meters, V1 = 1.5 meter per second and the density of water is 1000 kg per meter cube, then what will be the force. So first what we will do it, will solve the problems, will write these final expressions, then we will substitute the value.

Flow classification:

One dimensional

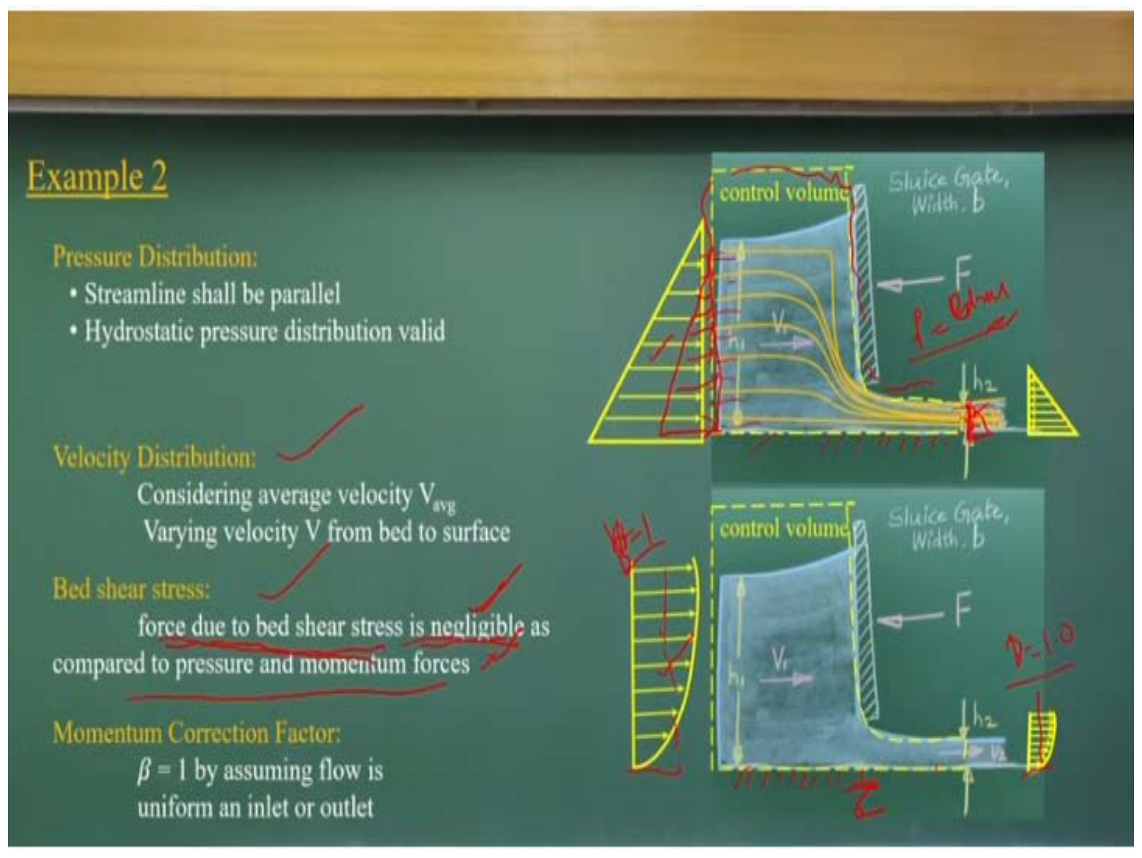
Steady

Turbulent

Incompressible

Now, we have to use a fixed control volume, that is what is there. So we will use a fixed control volume. Now, if you look it, in this control volume, will apply the pressure diagrams, force components, then we will have some adjustments to nullify some force component, then we will apply the mass conservation and linear momentum equations.

(Refer Slide Time: 43:10)



Now, first is pressure distribution. Now, if this is my control volume, first I need to draw the streamlines. So, as it expected that the streamlines will be like this, so as actual fluid valve if you have it could be like this, if the follow of actual fluid valve, if I follow it, it could be like this. So, we should consider the control volume such a way that, at a slow par distance such a way that all the streamlines should be the parallel, okay.

There is no curvature of the streamlines, where we have the control surface is cutting over that. If that is the conditions, if streamlines are parallel, then the pressure distributions of this area can be considered as a hydrostatic pressure distribution. That means as if flow is at rest conditions, whatever the pressure distribution that is pressure distribution that will happen when the streamlines are parallel. That means you define the control surface such a way that you can anticipate it in that region, the streamlines are the parallel.

The similar way, this outlet also we can consider it the streamlines are the parallel and I can consider the hydrostatic pressure distribution. Over the surface, we always can consider the pressure distributions equal to the atmospheric pressures. So, we use a gauge pressure concept to solve the problems. So, we need not to consider this atmospheric pressure distribution to compute the force, so, we can nullify that component. Second things this is what the pressure distributions.

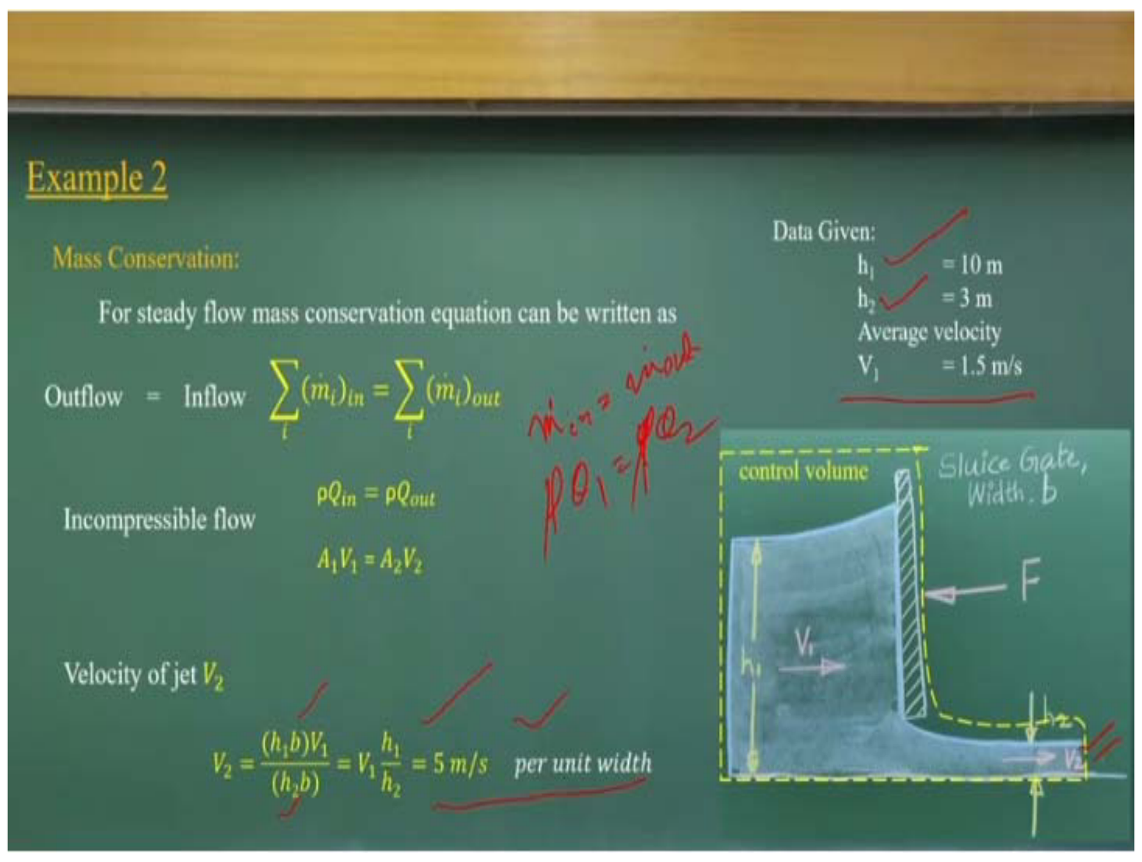
Now, you have to look the velocity distributions. As you expected that, here this velocity distribution will not be uniform, okay. The velocity distributions will be there, the zero velocity distributions at the wall and this value we get. So, in this case we consider is average velocity, okay. We do not consider the velocity distributions, okay. Or we consider is $\beta = 1$, the uniform velocity, but these are the assumptions, which is not valid in a real life problem, okay.

The velocity distributions will come like this. So, you have a uniform velocity distribution what is assumption is there and make it a $\beta = 1$ value, okay. So, this average velocity distribution is used and second thing that, at the surface which is connected to the wall, definitely there will be a shear stress acting on this. Because of this shear stress, there will be the force, but since here, the force due to the pressure distributions and momentum flux, rate of change of the momentum flux.

Those force are much, much higher order, than force due to shear stress. So, we can neglect it, as compared to the pressure and momentum force component concept. So, please remember it, there is a shear force acting on the bed, but in these problems, because the problem where you have these force components of the force due to the hydrostatic pressure distributions, the force due to the change of the momentum flux are much, much higher order compared to the force due to the shear stress at the bottom level.

So, to not to complicate the problems, we neglect it or we consider the force due to the shear stress is negligible as compared to the pressure and momentum force component. These are clear cut assumptions for velocity distributions and bed shear stress. Now, if you look it, as I said it, it is considered there is no velocity distributions, momentum is uniform in it outlet consider, these are all assumptions is there.

(Refer Slide Time: 47:27)



Data Given:

h_1 = 10 m

h_2 = 3 m

Average velocity V_1 = 1.5 m/s

Now I will apply mass conservation equation, because is single inlet and outlet conditions, the mass influx is equal to mass outflux, is a very simple problem,

Outflow = Inflow

$$\sum_i (\dot{m}_i)_{in} = \sum_i (\dot{m}_i)_{out}$$

$$\rho Q_{in} = \rho Q_{out}$$

Incompressible flow

$$A_1 V_1 = A_2 V_2$$

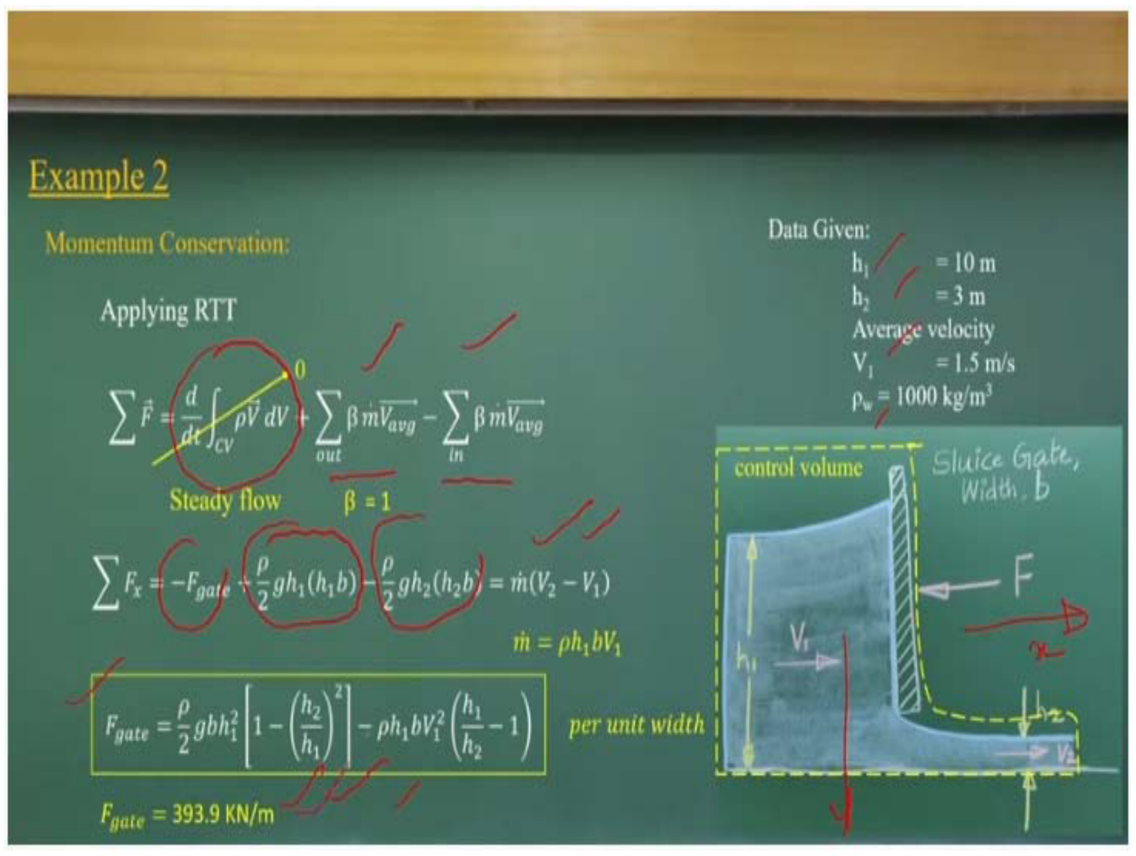
So, you know, h_1, h_2 you can consider a unit width perpendicular to the surface then the V will be cancelled out or you can V width, then you can compute the what will be the velocity. So, if I substitute, h_1 = 10 meters, h_2 = 3 meters, the average velocity what is coming 1.5 meters, as with depth of the flow is reduces, velocity increases, that the very basic conservation of mass talk about that.

Velocity of jet V_2

$$V_2 = \frac{(h1b)V_1}{(h2b)} = V_1 \frac{h1}{h2} \; = 5\; m/s \;\; per\; unit\; width$$

So, as the flow area decreases, the velocity increases that the basic idea, but it happens as a proportionality quantity, it happens 5 meter per second in this case.

(Refer Slide Time: 48:58)



Data Given:

h_1 = 10 m

h_2 = 3 m

Average velocity V_1 = 1.5 m/s

ρ_w = 1000 kg/m^3

Now if you look it, I will apply the conservations of momentum. Here, I am not simplifying the Reynolds transport theorem step by step. So, some of the force acting on this will be rate of the change of the momentum flux storage within the control volume, net outflux of the momentum flux, out in that part will be there, considering this β values.

Applying RTT

$$\sum \vec{F} = \underbrace{\frac{d}{dt}\int_{CV} \rho\vec{V}\, dV}_{0} + \sum_{out} \beta\, \dot{m}\overrightarrow{V_{avg}} - \sum_{in} \beta\, \dot{m}\overrightarrow{V_{avg}}$$

So, in case of the steady flow, I can make it that is 0 and $\beta = 1$, that is what we have discuss it. And if I am applying the force components only in these directions, okay.

$$\sum F_x = -F_{gate} + \frac{\rho}{2} g h_1 (h_1 b) - \frac{\rho}{2} g h_2 (h_2 b) = \dot{m}(V_2 - V_1)$$

$$\dot{m} = \rho h_1 b V_1$$

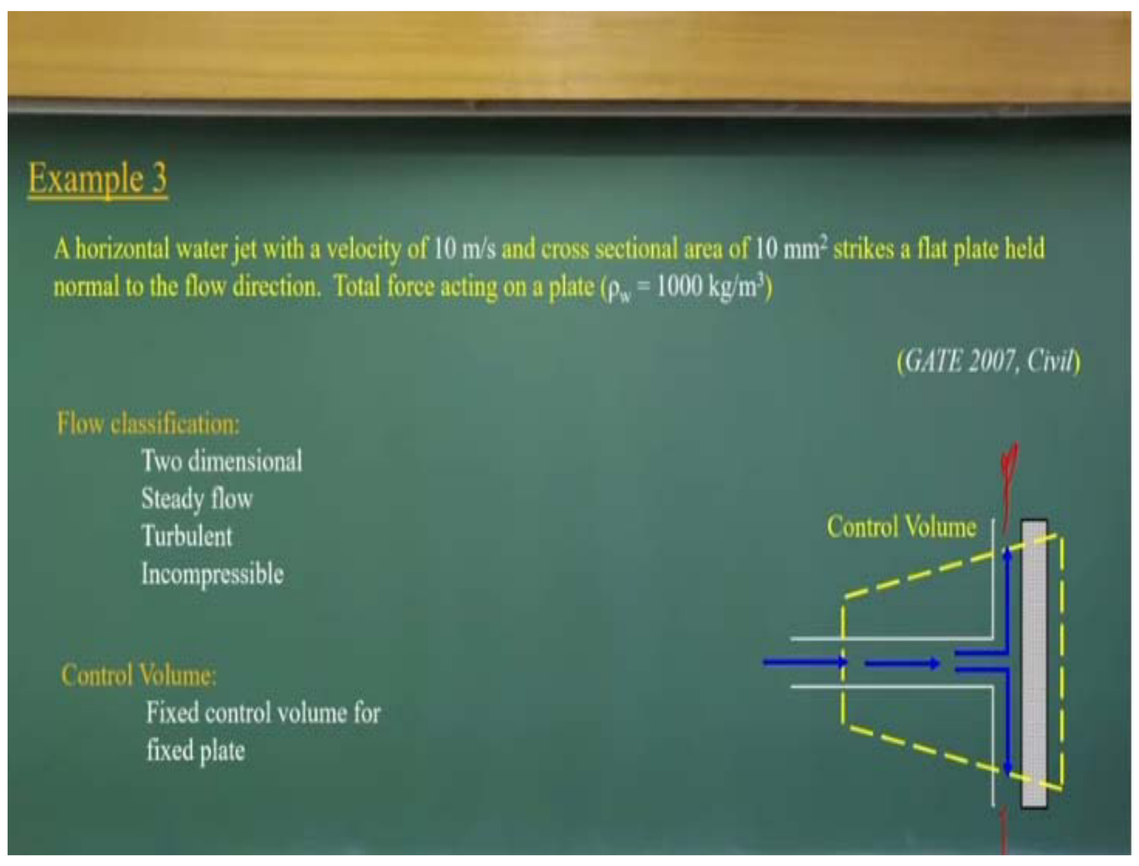
Not in the X directions, my body force is acting on the Y directions or the Z directions. So I will have a force gate is equal to, this is the pressure force component at these locations equal to rate of change of the momentum flux, the mass flux will be the same, b2 – b1, that is what will be the rate of change of momentum flux and if I substitute it and solve with this equations, I will be get it like this, very simple way. This is what the expressions will come it in terms of h1 and h2, b1 and the densities row.

$$F_{gate} = \frac{\rho}{2} g b h_1^2 \left[1 - \left(\frac{h_2}{h_1}\right)^2\right] - \rho h_1 b V_1^2 \left(\frac{h_1}{h_2} - 1\right)$$

$$F_{gate} = 393.9 \text{ KN/m}$$

Then, I will substitute the values as it is given here, v1, v2, then I will get it, the force acting on these will be 393.9 kilo Newton per meters. So, this much of force will be acted because of the flow what is coming it, what is the rate of change of momentum flux and the pressure force difference, that what will exit, will be force on the gate, that what will be get it from this case.

(Refer Slide Time: 50:55)



[A horizontal water jet with a velocity of 10 m/s and cross sectional area of 10 mm^2 strikes a flat plate held normal to the flow direction. Total force acting on a plate (ρ_w = 1000 kg/m^3)]

Flow classification:

Two dimensional

Steady flow

Turbulent

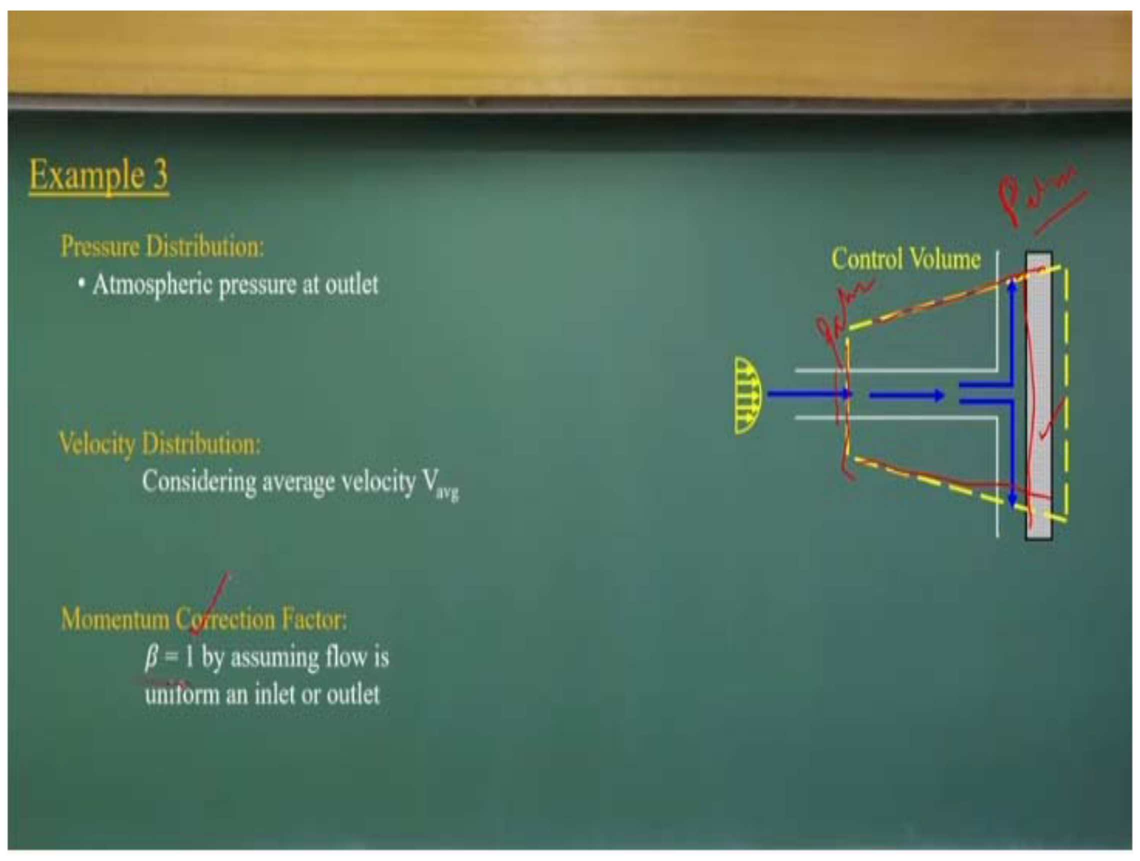
Incompressible

Now, let us consider another problems, which is a GATE 2007 civil engineering course problems. Is very simple problems, there is a horizontal water jet with a velocity 10 meter per second, cross-sectional area is 10 millimeter squares, strike in a flat plate, held normal to the flow direction, what is the total force acting on the plate? That what is the questions. If you look at that, again we have to do flow classification. As is expected, as you see, these diagrams, flow has to two directions, okay two dimensional.

The jet force what is coming in the X directions, after hitting, its moving in the Y direction splitting into two part, moving in the Y directions. If it is having symmetrical water jet, the same amount of water will go, in this direction and in this direction, because the gravity force will be much lesser component as compared to the rate of change of momentum flux, the force component is much, much larger than gravity force. There will not be imbalance between these two velocity components.

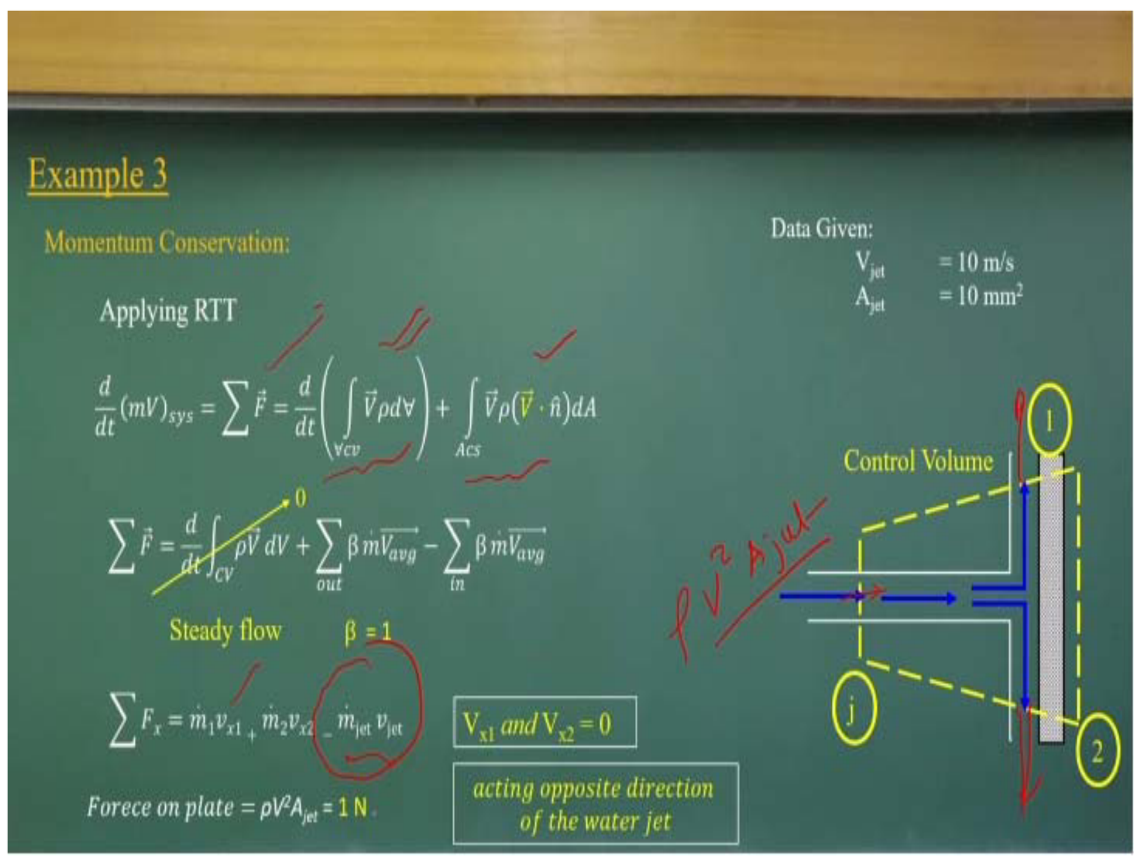
So more or less, the same amount of the flow will go from this direction and this directions. So, that way, what our jet is we gained that, we can consider a fixed control volume.

(Refer Slide Time: 52:22)



Once you consider the fixed control volumes like this, then you apply the pressure distribution, if you remember it that in this water jets are into the atmospheres, always you can use the pressure distribution is atmospheric pressure. So, over this control surface, I can find out the atmospheric control surface, okay. Only the force what will be impact on this, what we need to compute it. Here, also we have consider that there will be a velocity distribution over the water jet. But we consider the velocity distributions is uniform that $\beta = 1$.

(Refer Slide Time: 53:04)



Assuming that, we apply this momentum conservation equations, okay. And it is a very simple things, momentums influx will be the momentum outflow. And if I apply it, again this Reynolds transport theorems as basic equations, some of the force acting on this control volume, that will be rate of change of the momentum flux to its within the control volume or net outflux of the momentum flux, passing through this control surface.

$$\frac{d}{dt}(mV)_{sys} = \sum \vec{F} = \frac{d}{dt}\left(\int_{\forall cv} \vec{V}\rho d\forall\right) + \int_{Acs} \vec{V}\rho(\vec{V}\cdot\hat{n})dA$$

That what if I put it and take this assumption are steady problems and flow is impacting the X and Y directions and if I result the velocity components in there, three component is one, two, and the jet is with the jet directions and these. So, some of the force acting on this will be the momentum flux x directions, which is exiting from this momentum flux these directions exiting out and what is the momentum flux coming into this one. But as you remember, this two velocity component of this are the 0.

$$\sum \vec{F} = \underbrace{\frac{d}{dt}\int_{CV} \rho\vec{V}\, dV}_{0} + \sum_{out} \beta\, \dot{m}\overrightarrow{V_{avg}} - \sum_{in} \beta\, \dot{m}\overrightarrow{V_{avg}}$$

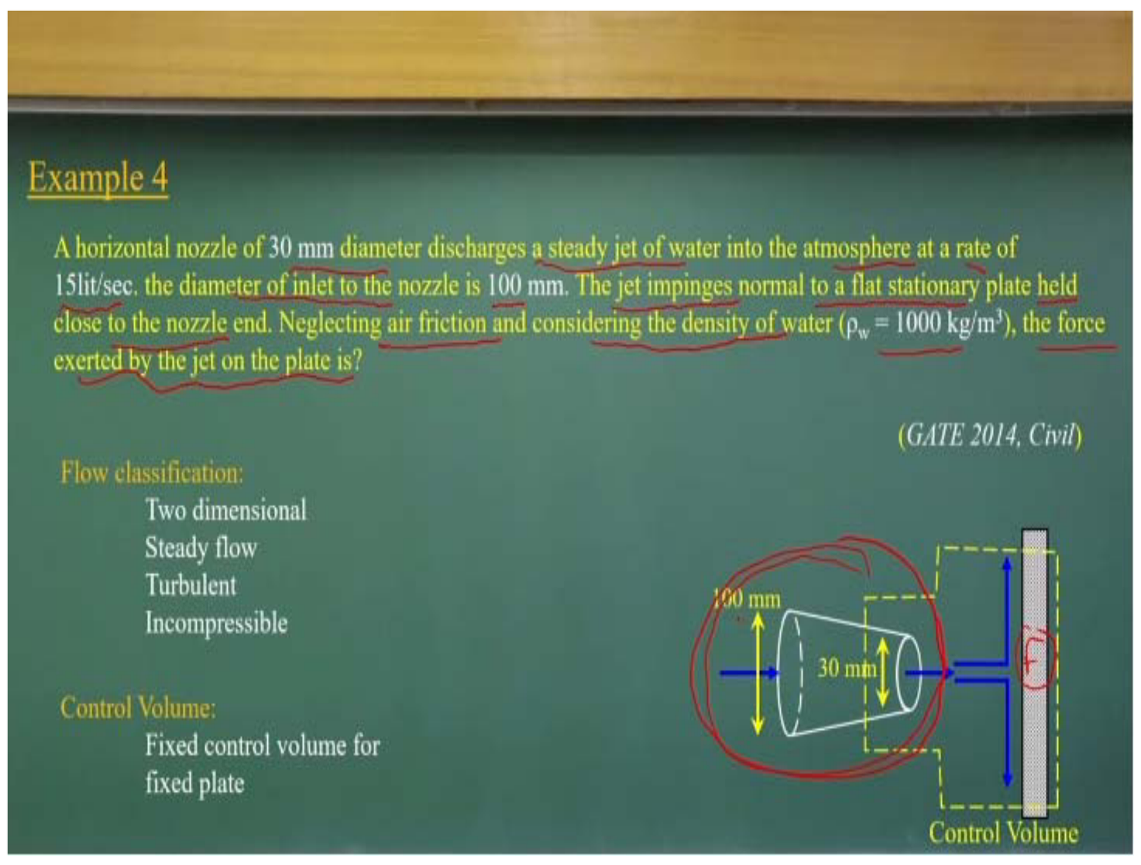
$\sum Fx = \dot{m}_1 v_{x1} + \dot{m}_2 v_{x2} - \dot{m}_{jet}\, v_{jet}$

V_{x1} *and* $V_{x2} = 0$

$Forece\ on\ plate = \rho V^2 A_{jet} = 1$ N

acting opposite direction of the water jet

(Refer Slide Time: 54:46)



Similar way the same problems get 2012, only this additional things added to here, okay. The same jet is there, but there is a horizontal nozzle 30 meter diameter, discharge steady jet into the atmosphere at the rate of 15 liters per second. The diameter of inlet to the nozzles is 100 millimeters, okay. That is what, the 100 millimeters to the 30 millimeters, the jet impinge on a normal to a flat stationary plate, held close to the nozzles and neglecting air frictions considering the density of water is equal to 1000 kg per meter cube.

[A horizontal nozzle of 30 mm diameter discharges a steady jet of water into the atmosphere at a rate of 15lit/sec. the diameter of inlet to the nozzle is 100 mm. The jet impinges normal to a flat stationary plate held close to the nozzle end. Neglecting air friction and considering the density of water ($\rho_w = 1000$ kg/m^3), the force exerted by the jet on the plate is?]

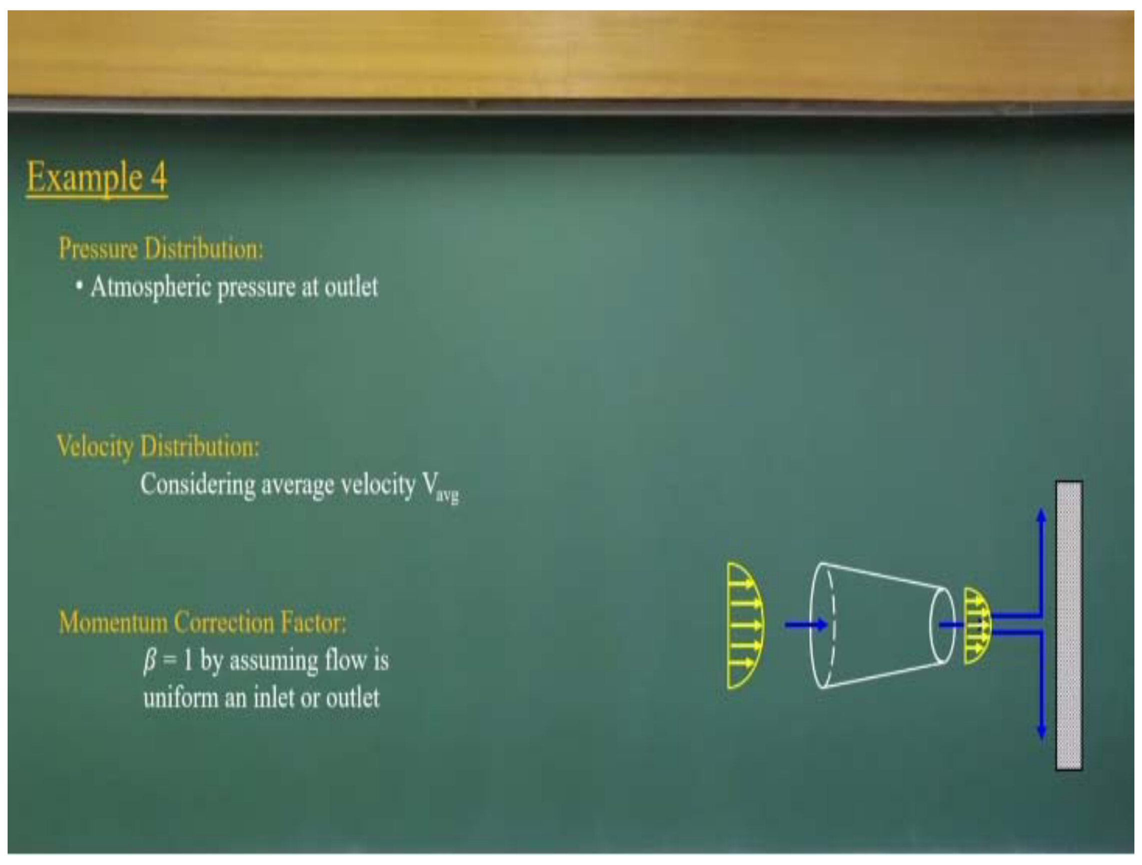
Flow classification:

- Two dimensional
- Steady flow
- Turbulent
- Incompressible

What could be the force exerted jet on the plate is? This problem exactly the problems of the example three, only these components there, there is a one nozzles is there, which is having the reducing the diameters from 100 millimeter to 300 millimeters, because of that there is a change of the velocities and that velocity impairs here to find out what is force acting on this. The problem is exact similar problems what we discuss in example three, only there is a nozzle is there.

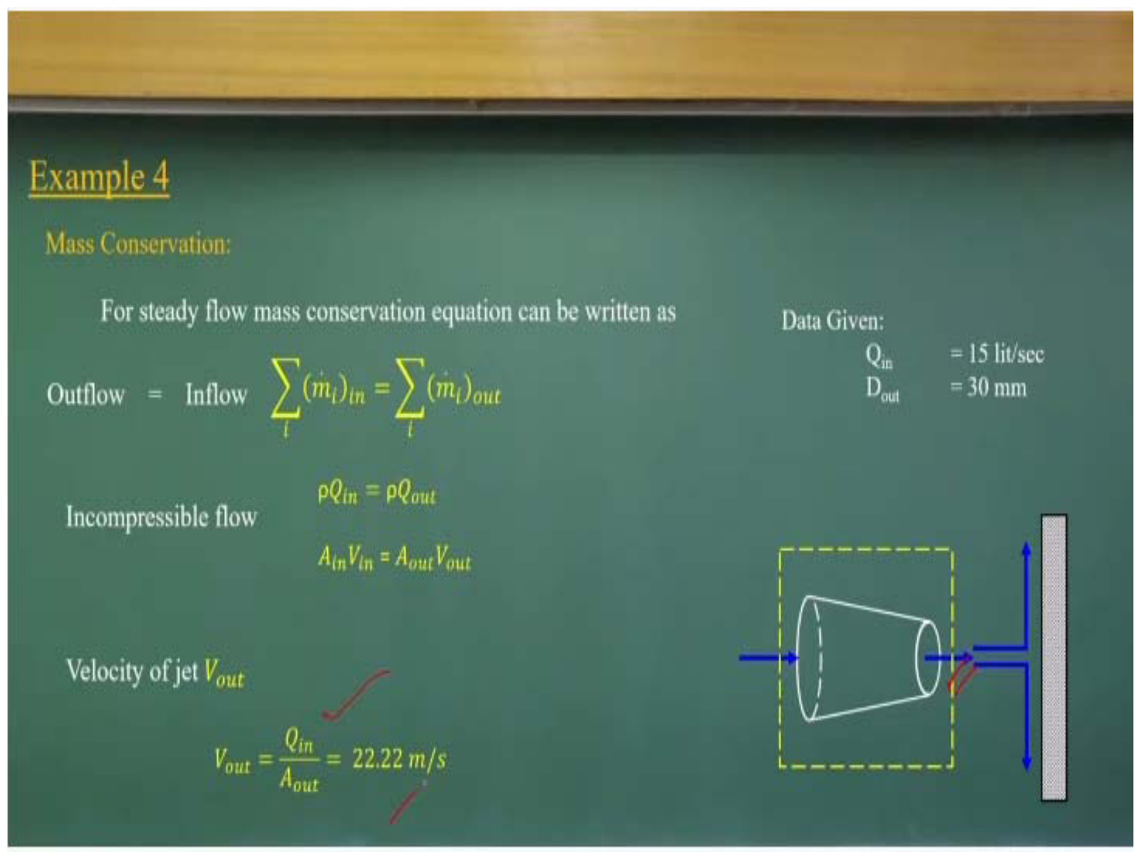
So again, the flow classification is two dimensional, steady flow, turbulent and incompressible, fixed control volume.

(Refer Slide Time: 56:24)



And will have a, pressure distributions will consider all this atmospheric pressures, average velocity concept will use it, as you see this velocity distribution will be come it like this, not the average velocity or the uniform velocities, but to solve these problems, we consider is average velocity and β, corrections factors equal to the 1.

(Refer Slide Time: 56:48)



And if it's that, now I apply mass conservation equations, first at the nozzle levels, inflow is equal to the outflow,

For steady flow mass conservation equation can be written as

Outflow = Inflow

$$\sum_i(\dot{m}_i)_{in} = \sum_i(\dot{m}_i)_{out}$$

$$\rho Q_{in} = \rho Q_{out}$$

that is why Incompressible flow

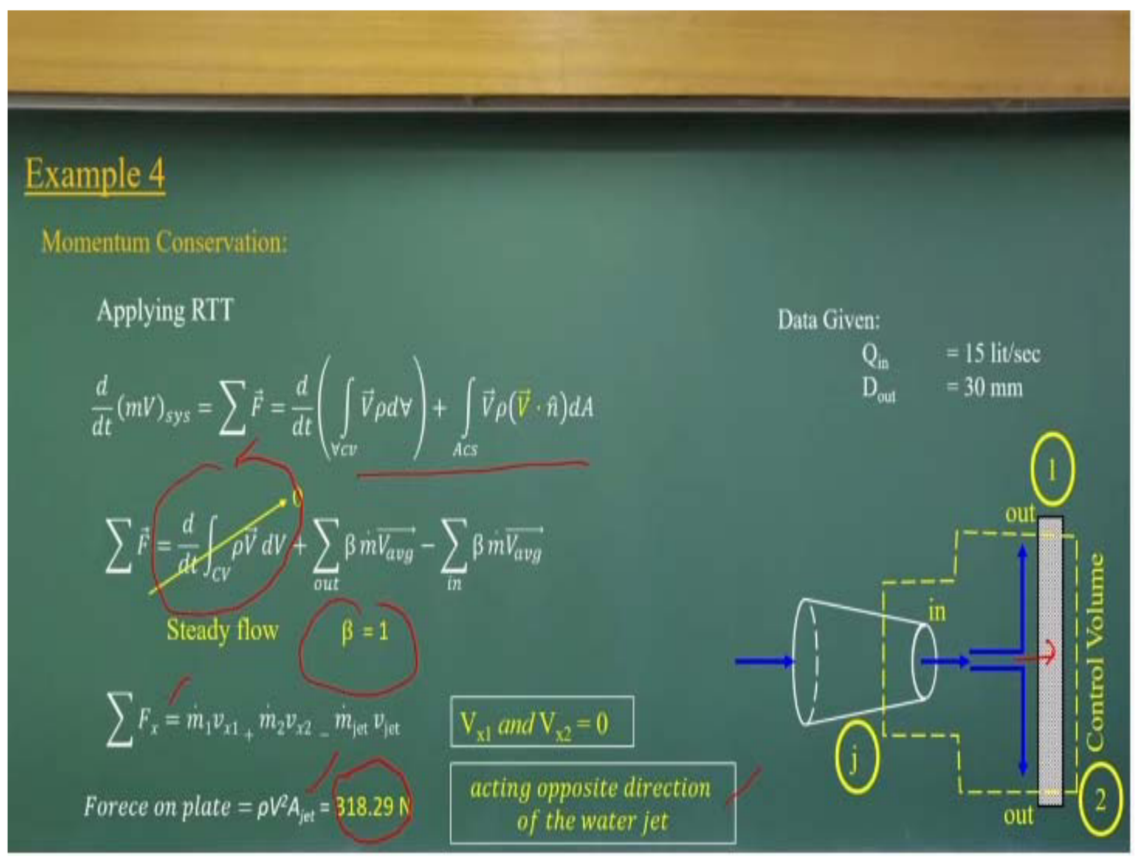
$$A_{in}V_{in} = A_{out}V_{out}$$

Velocity of jet V_{out}

$$V_{out} = \frac{Q_{in}}{A_{out}} = 22.22\ m/s$$

This is V out from that, because it reduce the diameter of the nozzles, you will have more of the velocities, that is the concept what is there.

(Refer Slide Time: 57:38)



And now, you apply the momentum equations, momentum conservation equation we can also apply it, for this control volumes.

Applying RTT

$$\frac{d}{dt}(mV)_{sys} = \sum \vec{F} = \frac{d}{dt}\left(\int_{\forall cv} \vec{V}\rho d\forall\right) + \int_{Acs} \vec{V}\rho(\vec{V}\cdot\hat{n})dA$$

0

$$\sum \vec{F} = \frac{d}{dt}\int_{CV} \rho\vec{V}\, dV + \sum_{out} \beta\, \dot{m}\overrightarrow{V_{avg}} - \sum_{in} \beta\, \dot{m}\overrightarrow{V_{avg}}$$

Another control volume we have considered, when a jet of impact is loading it, then we write the Reynolds transport theorem here, force is equal to this components. Then, we simplified this becomes 0, because of steady flow, $\beta = 1$ and again having the same component of X direction, force component if you are talking about, which meter is your competing wall.

$\sum Fx = \dot{m}_1 v_{x1} + \dot{m}_2 v_{x2} - \dot{m}_{jet}\, v_{jet}$

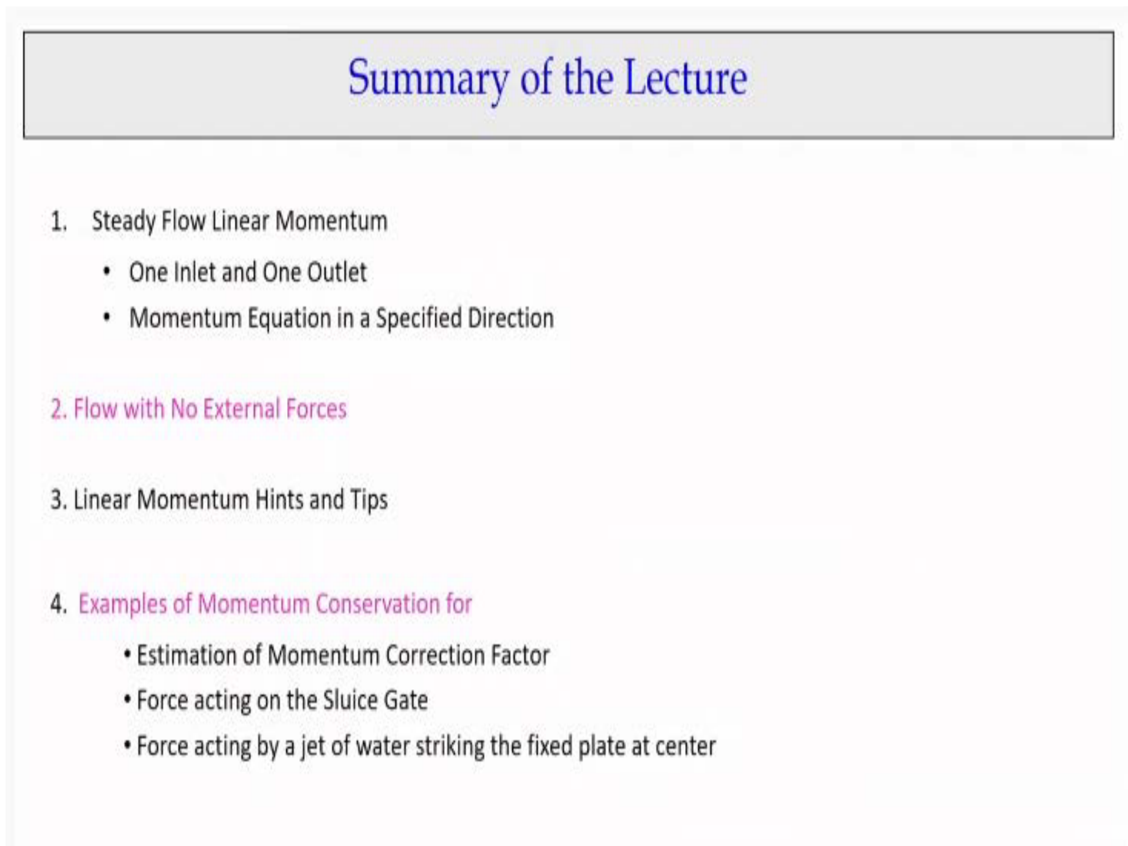
V_{x1} *and* $V_{x2} = 0$

$Forece\ on\ plate = \rho V^2 A_{jet} = 318.29$ N

acting opposite direction of the water jet

that much of force reacting it. So this way, we have solved the four problems. Let me summarize the problems that.

(Refer Slide Time: 58:47)



We discussed how we can simplify the linear momentum of equations for one inlet, one outlet come it. Momentum equations we can apply for a specific direction, instead of vector equations we can do it. We also discussed that linear momentum equation you can apply where no external force since exists. So we can do it, then I discuss very thoroughly whenever you draw the control volumes and try to find out the momentum flux, you follow certain hints and tips.

And if you follow the hints and tips, you can simplify a complex problems and you can apply appropriate control volumes. Applications of the appropriate control volume is art, like a free body diagram, you apply for solid mechanics. Similar way, you should have solved the many problems using the control volume concept. As soon as you see a problems, you do a flow classifications and find out appropriate control volumes and those control volumes, the surfaces and all should follow the hints and tips, what we discussed linear momentum equations.

No doubt we discuss about four different type of problems to how we can apply linear momentum equations and the mass conservation equations. The next class also will discuss more example problems on these with a moving control volume concept also. With this, let us conclude this class. Thank you.