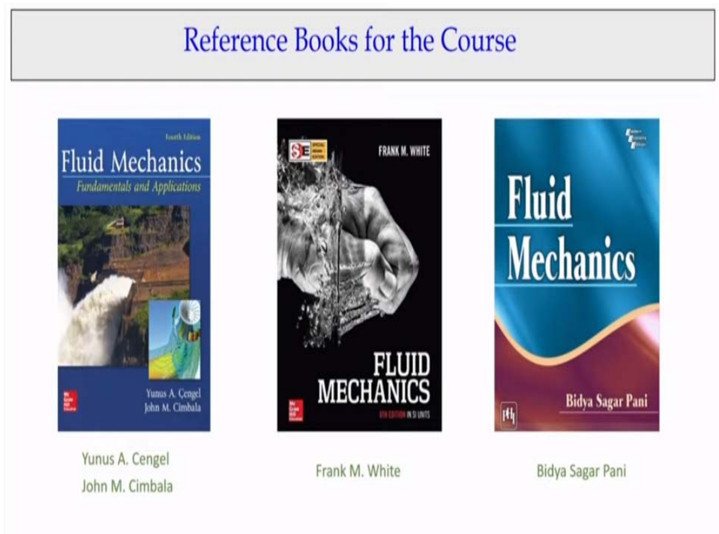
Fluid Mechanics for Civil and Mechanical Engineering
Prof. Subashisa Dutta
Department of Civil Engineering
Indian Institute of Technology Guwahati

Lecture No 10
Conservation of Momentum and its Applications

Welcome to you this course on a fluid mechanics. It is a very interestingly that today we have a 10th lecture, which is the half way of the fluid mechanics course, what I have been teaching you and today will cover the conservations of momentum and its applications, which is really a interesting subject, in the fluid mechanics using the Reynolds transport theorems and the control volume concept.

(Refer Slide Time: 01:04)



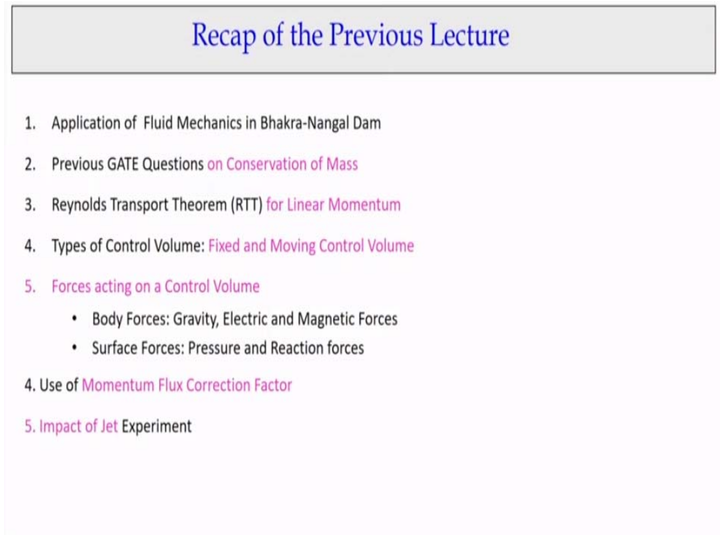
Considering that aspect, I will go through you how we can simplify these Reynold transport theorems for a control volumes and how we can simplify in terms pressure distributions and the velocity distributions. So, how we do the simplifications of these equations and then we apply that equations for, a engineering applications like finding out the force components, the velocity component and the pressure distributions.

As I stated earlier, I just follow the books of these three, the mostly I am talking about a mid-path between fluid mechanics books by FM White, which is big mathematical oriented, whereas if you talk about fluid mechanics fundamental and applications, which is more illustrated oriented. So, I tried to make it to you in between of these mathematics and illustrations oriented and other book as you know it, this fluid mechanics by Professor Bidya

Sagar Pani.

So, let us come back to the recap of the previous lectures. As we said it earlier, the basically the mass conservations equations.

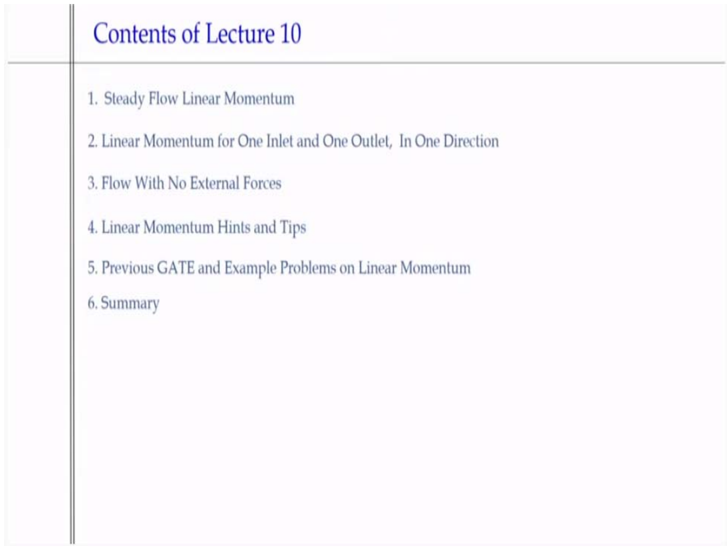
(Refer Slide Time: 02:18)



And we also solved the GATE questions on mass conservation equations. Then we also discuss very details that the Reynolds transport theorem for linear momentum equations and that is what are the two control volumes; one is fixed control volume another for the moving control volume. And that what we talked about the force acting a control volume. There will be two types of forces, body forces and the surface forces. The surface forces are pressures and the reactions forces, that what we talked a lot.

Discuss also today body forces mostly what the problems we can consider is only the gravity force we consider as a body force, not the electric magnetic forces. Then, we also discussed about the momentum flux correction factors, this is a very simplified concept used to determine the momentum flux passing through a non-uniform cross-sections and using this correction factor, momentum flux correction factors. And also I gave you a example of jet experiments, how we conducted in laboratory.

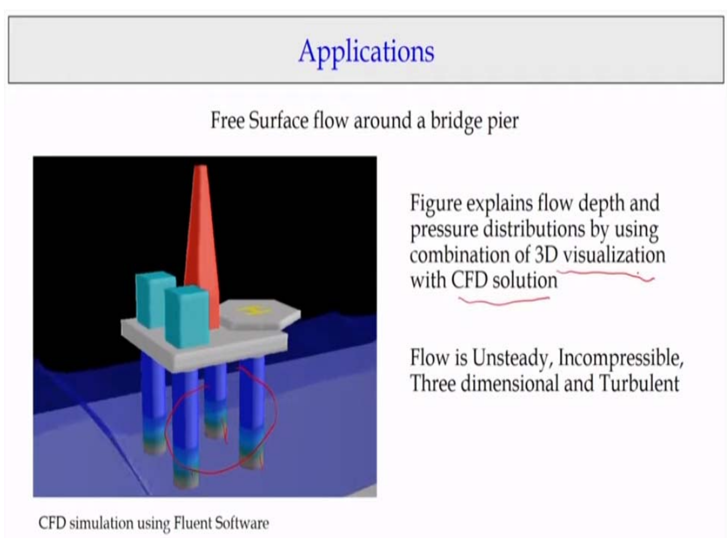
(Refer Slide Time: 03:34)



Now let us come to today's lecture, will have a steady flow across missions for the linear momentum equations. Many of the times, we have a linear momentum equations we solve for the one inlet or one outlet and in one directions and some of the problems we have solved with no external forces, then how the momentum equations can be simplified. Then fourth what we will talk about, when you apply the linear momentum equations, what are the hints and tips, what should we consider when you apply that linear momentum equations.

Then we will commit to solve for example problems of previous GATE questions, we will solve it, then we will have a summary of today lectures. Now let us come back to very interesting 3D figures what you can see it.

(Refer Slide Time: 04:28)



It is, there are bridge piers, there are the bridge piers are here and the flow is coming it, which is unsteady flow. And if you look at this color fringes, its showing how the pressure diagrams are changing, just you look it, how the 3D view we were getting it, how the flow is passing through a bridge piers and how the pressure distributions are changing it, if you look at these colors of this ones, how the pressure distributions are changing it. This is today's world, is a possible, because the computational fluid dynamic solutions are available as well as the 3D visualizations tools are developed.

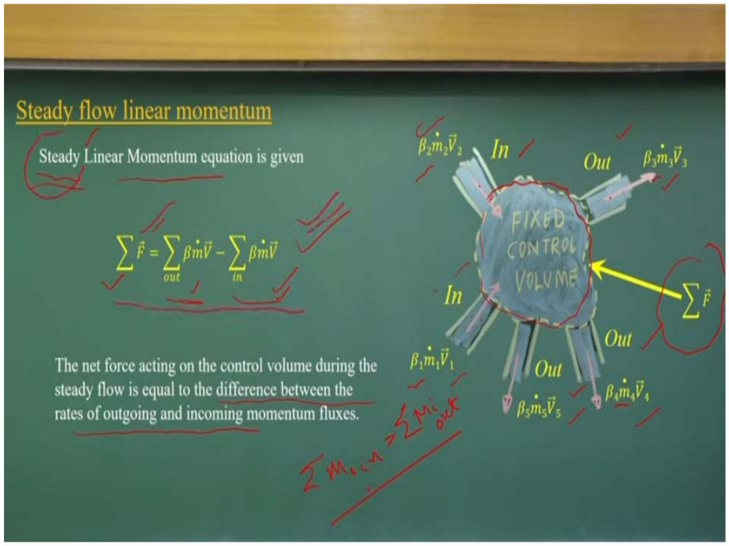
So, you can feel like that we have conducted an experiment in a fluent to get it how the pressure distributions, how the flow of the various sensors happening it, in a group of bridge piers are there and how they are interacting each other, if you can look at these 3D figures. If you look it this way, this flow is unsteady, is perish with the time, incompressible flow, 3 dimensional and the turbulent flow. So, all the complex flow we can solve it using the CFD and we can visualize that to in 3D visualizations.

Somebody can feel it, how the flow is going on. If you can look at that, there is a breaking of the waves is also happening it, if you just to look it, there is a breaking of the waves are happening it and how this flow is coming it, how the hitting in different bridge piers and the different bridge piers having the different amount of pressures acting on the force due to the pressure diagrams are the different. So these types of visualizations nowadays it is possible with help of the CFD solution the 3D visualizations.

It is possible nowadays to solve the full fledge turbulent Navier-Stokes equations with some aphorisms, it is possible. So that is why pointed to tell that even if I just talk about the control volume concept, which is a very gross characteristic, but today it is a possible to have a this to 3D visualizes and the CFD solutions which gives us, just a real life problems like what I am showing it, a pre-surface flow around the bridge, the breaking of the waves, and also the pressure distributions are changes from the pier to pier.

All its a possible because of the CFD solutions of have three dimensional turbulent unsteady incompressible flow, that is what is available, also 3D visualizations. With this note, let us go to the our level where we are talking about the integral concept of control volumes what we apply it for a simple problems.

(Refer Slide Time: 07:19)



Now, go for a simple problems, where you have a linear momentum equations like you have a fixed control volume, just to look at, you have a fixed control volume. There are the in are there, okay, and there are the outs are three, and this is a fixed control volume. So there is inflows, this is outflows. There is inflows and outflows. The sum of the force will be act on these, if it is a steady linear momentum equations.

$$\sum \vec{F} = \sum_{out} \beta \dot{m} \vec{V} - \sum_{in} \beta \dot{m} \vec{V}$$

The time derivative components become 0, so we will have a sum of force is equal to, this is still I am putting in a vector notation is equal to the, in case of steady, the difference between the rate of outgoing and incoming momentum fluxes. So what we will do it, some of the momentum flux, what is coming as an inflow, that the mass flux into the velocity that is a momentum flux, then we have the beta one and beta two, which is representing momentum flux correction factors, which will differ from the one to 1 to 2 3 4 5 locations because the velocity distributions are the different.

So, as the velocity difference are there, so you will have a B_1, B_2, B_3, B_4, B_5 will be the difference because the momentum flux correction factors, it depends upon the velocity distribution. If when you have velocity distribution uniform, the beta one will be the one but if its not uniform, the beta one value will be the different. So, the basically if you look it to apply this concept, we use velocity distribution as a, considering the velocity distribution.

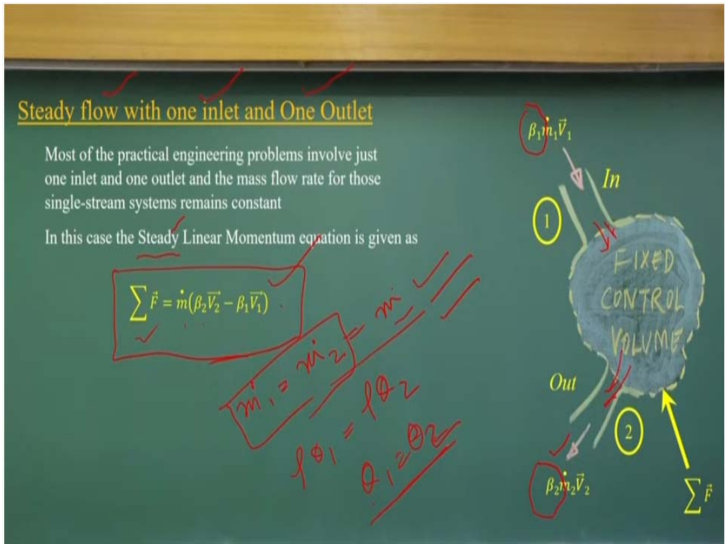
So we consider momentum flux correction factors, we have mass rate come into the velocity which is the momentum flux rate that what we are summing it how much in and how much going out from this. Incoming momentum flux and outgoing momentum flux. As you will know from factorically, so outgoing momentum flux will be the positive and incoming momentum flux will be the negative. So, the rate of change of this net out flux of this momentum flux will be the force component.

The sum of the force component to what this control volume that what will be the net out flux of momentum fluxes through these control surface. So, in this case, we have three outlets, two inlets. So for the two inlets, we can compute the momentum flux from 1, 2. Similar we can compute the momentum flux, which is going out 3, 4, 5 we can compute it. Also we can consider, the momentum flux correction factors, which will be the different for the different inflow or outflow flow distributions, velocity distributions concerned.

So, we have considered that fact that there is a velocity distributions variability is there. Mass flux lean and out differently will be there. So considering this, in these control volume, fixed control volume, what will be the net force which will be acting on these will be considered as a steady problem, then you will have these conditions, okay. You can easily find out if it is a steady problem, you will have a definitely the net mass influx, in should equal to the net mass influx going out from this, that should be we get.

So, whatever the sum of the mass influx is coming it, that should be equal to some of the mass outflux is going out from this control volume, then we have steady problems. So, this is the conservation of mass equations and this is the linear momentum equations for the steady flow having multiple inlet and the outlet.

(Refer Slide Time: 11:30)



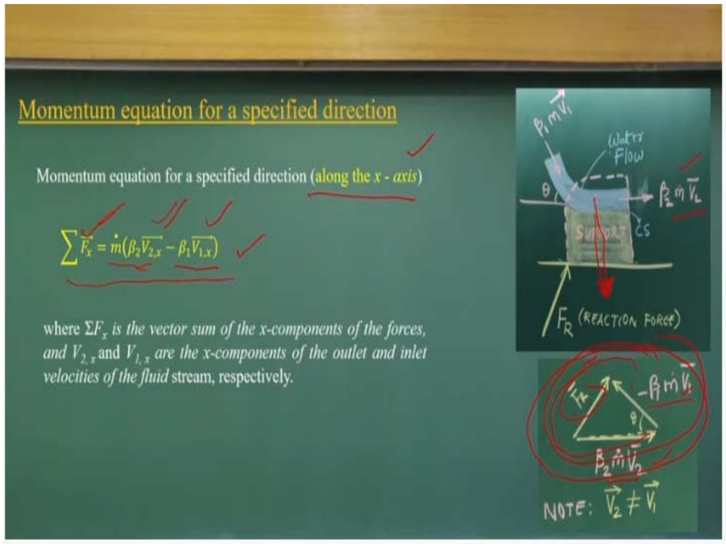
Now, let us go for another simplifications. That in a steady flow conditions, okay and only one outlet one inlet, this is very simplified case. See when you have one inlet and one outlet, if your mass influx is $\dot{m}_1$ is inflow is coming into that, your mass influx out will be $\dot{m}_2$, that should be equal. That means, $\rho Q_1 = \rho Q_2$ will be equal that means Q1 = Q2, that is a very basic equations what we get it, if we apply the conservations of mass.

So, if mass influx, what is coming if that equal to mass outflux, that what we can designate as a mass influx rate and then, the sum of the force acting on this as a steady conditions what will happen it, that some of force equal to the mass will come out β_2 P_2 – β_1 P_1. So, outlet inlet momentum flux what we are getting it with a correction factors of β_1 and β_2 we have use it to compute the momentum flux, what is coming into the control volume, what is going out from this control volume. The net momentum flux, that what will be the force component.

$$\sum \vec{F} = \dot{m}(\beta_2\overrightarrow{V_2} - \beta_1\overrightarrow{V_1})$$

That is the equations what you get. So, these simplifications what we are doing it, that sometimes you can directly use it, these equations instead of to use Reynolds transport theorems, then you simplify it, then you come it. Instead of that, if you have solving a steady flow problem with one inlet and one outlet, to find out what is a force is acting in that, you can directly substitute these equations as a linear momentum equations and these equations you can use as mass conservation equation, which is the Q1 = Q2. That is very simple way we can do it.

(Refer Slide Time: 13:36)



Now, there are the problems where is come it, very interestingly that we can apply the momentum equations as you can go it, this momentum equation is a vector equation, it has a three components in scalar directions, X component, Y component, Z component. Many of the time, we result at specific directions, in this case is X directions to find out the force components. That is what we try to look it, what will be the force is happening it, in the X directions only.

$$\sum \overrightarrow{F_x} = \dot{m}\left(\beta_2 \overrightarrow{V_{2,x}} - \beta_1 \overrightarrow{V_{1,x}}\right)$$

So, we can use it only the X directions component part, since there is, this problem if you take it, there is a water jet is coming in and going out, there is a change in the directions of the hot water jet okay. And there is a support system for that, what will be the force acting on this, that is what the problems. That means there is a water that is coming which is making an angle θ, then making a turn it then becomes a horizontal water jet. So, because of the change of the momentum flux directions, what is the amount of force is acting on that.

If I am resumed it only this X directions component, then what I will do it, I will find out the momentum flux in X direction, not the Y or Z direction, because I can have a three equations. Here, we have a very good simplification that, if I am to compute the force component only the X directions, if I can see that the control volume the slide this, I know it the force, the gravity force act in the Z directions. So that force component will not commit. So, this is what quite simplifications we supposed to do it when you consider only the X direction.

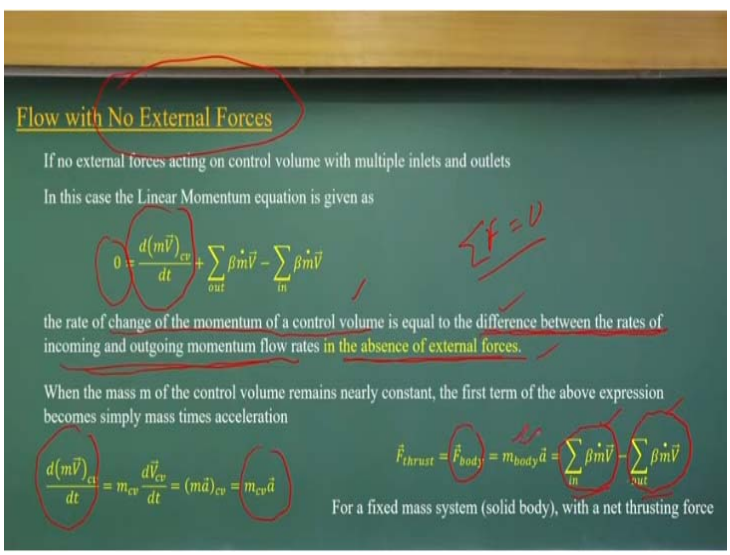
The force component of X directions, you know it the force acting on the, either in Y or the Z direction, that may not have any components. So, that way you can result the force component as a scalar component only X directions. So you compute the momentum flux what coming in the X direction momentum flux, similar way what is a momentum flux outgoing, that what we compute it. Sometimes you can exactly a vector method we can prove it if you do a vectorically.

The momentum flux in and out that can vectorically, that what is the momentum flux is a force components, so you can result this force component and the resultant force you can get. That is what we can simplify, you can solve this directly that these are the momentum flux as equivalent to force what is coming onto this control volume, going out from this control volume, you know these two force balance with a resultant forces.

That what vectorical you can find out what will be the F_R value, that what will be the reaction forces. So, this way we can also simplify either considering this control volume or writing the force factors due to the momentum blocks in and the out, from there we can get the result and force components and we can get it the what is the x exponent force component. So note it that, B2β_2 should not equal equal to P_1, if it is that, so that because net force will be the 0.

So okay, there is no change in the velocity, directions or the magnitudes then there is no force which is going to act on this.

(Refer Slide Time: 17:06)



Now if you look it there another set of the problems we solve it, where there is no external forces is there, there is no external forces is there, very simple cases. So, in that case, if I use a linear momentum equations, the sum of the force will be the zero.

$$0 = \frac{d(m\vec{V})_{cv}}{dt} + \sum_{out} \beta \dot{m}\vec{V} - \sum_{in} \beta \dot{m}\vec{V}$$

So what will happen it, that rate of change of momentum on a control volume that is what is here, that is which is equal to the difference between the rate of the incoming and outgoing momentum flux rates in absence of external forces.

If there is a, you consider a control volume, where we visualize conceptually that there is no external forces acting it. So, that problem is quite simplified, because if you apply the linear momentum equations, the net force is equal to zero. So zero what we have applied. So, problems becomes rate of change of momentum within the control volume is equal to the difference between the rate of incoming, outgoing momentum flux in absence of forces.

So, we have these three components with these if you can simplify it is a momentum rate what it changes is that what will be the as equivalent is mass of control volume and the acceleration component, it is just like a rigid body is moving it, okay. So, that what will be the force is moving it MD, that what will be, that force will be equate with the difference between the momentum flux in and the out. That the conditions will come it.

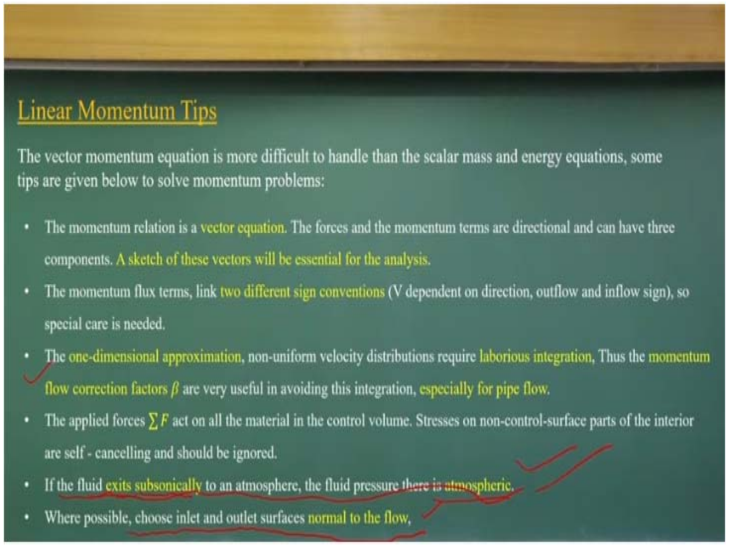
$$\frac{d(m\vec{V})_{cv}}{dt} = m_{cv}\frac{d\vec{V}_{cv}}{dt} = (m\vec{a})_{cv} = m_{cv}\vec{a}$$

The momentum flux in and out will be there, the sign convention if you look it, when you have a positive and negative, just have the difference equations of we are equating it. So, you can find out in a systems net flux, momentum flux is what is going on in and out, that what will be mass of bodies, M body and has accelerations that what will be equated that. If there is no external force acting on that or net thrusting force will be fixed mass body will be that much,.

$$\vec{F}_{thrust} = \vec{F}_{body} = m_{body}\vec{a} = \sum_{in} \beta \dot{m}\vec{V} - \sum_{out} \beta \dot{m}\vec{V}$$

This mostly we use it this equations for when you were launching the rockets, the net momentum flux what will you get it, the accelerator to the body. That the problems we solve it, I will solve one of the problems based on this concept.

(Refer Slide Time: 19:39)



Now comes back to the, what are the tips are there, when you apply the linear momentum equation. First thing is that do remember is momentum equations is a vector equation, it has a three scalar components, it has the directions. You can write it in X direction equations, Y direction equation and the Z direction equation. You can write it in the three directions. So, many of the times, you do not result into three directions, we result the equations only one direction to solve a particular problem, okay.

But we can resolve this momentum relation in three directions X, Y, Z. So, when you solve this, any linear momentum the questions please case the vector components, then you visualize the fluid flow problems and then you solve it. Because it is a vector equation, you try to apply as a vector equation. Second is that we always compute the momentum flux terms, in flux and out flux terms. You always look it, what is a velocity direction. Whether it is a inflow or the outflow.

What will be the conditions when you have a dot product or the scalar product of the velocity and normal to the control surface, whether it will be a positive sign or a negative sign. That what always you visualize or always try to draw the velocity diagram first. Then, you have a control surface, you try to find out, if I multiply the scalar product of velocity and the normal vector to the surface, whether it will be the positive sign or the negative sign. That means if $\theta = 0$ between the velocity factors and the normal vectors.

Then we have cos theta equal to the 1. Or if you have a $\theta = 5^{\circ}$, the $cos 5$ will be the -1. So, that what you try to look at that, what is the directions of velocity vectors, what is the directions of the normal vectors to the control surface? If I do scalar product of the velocity and the normal vectors, what is the side, whether either positive or negative side. Many of the time, the student do these mistakes that they have a confused that whether you have to use a positive or negative sign.

Please find, try to understand it you are making a scalar product of velocity vectors and the normal vector. So, if you know what is the angle between these two, if $\theta = 0$ or $\theta = 5^{\circ}$, you can find out whether it is a positive or the negative, this is very simple thing. But you guage the velocity factors, guage the normal factors then you apply that. And one of the things what we always discuss is that whenever you take a problems, the velocity distributions is not uniform.

Any real life problems, velocity distributions are not uniform, but we simplified it, make use of momentum flux correction factor. which considered the, because of non-uniform velocity distributions, what could be the correction factor to compute momentum flux through a control surface? So, that way, we always try to look it that, there is always a momentum flux correction factor will be there, if there is some velocity distributions is not uniform. For a real life problems velocity distribution cannot be uniform.

And that is the conditions we always have with the beta value or momentum flow correction factors. So, in a problem if the momentum flux correction factor is not given it, you can assume it or highlight it that, what you have done it that you have assume it that momentum correction flow corrections factor is 1 or unit file. That is what you have to consider, it is very important things about momentum flux correction factors and students who visualize that, there is a flow distribution the velocity distribution.

That what is considered when you use the momentum flux correction factors. And second things what is there, this applied force acting all the material in the control volume, we do not bother about inside the control volume, how the no control surface, the force acting part, that what self-canceling each other's, that what we do not consider it. We talked about over the control surface, what are the forces is acting it, that what we consider it.

And the last one is what is that the fluid when it exists subsonically, that means flow is subsonic flow, Mach number is less than 1. In that case, if flow exits to atmospheric conditions, you can always consider the fluid pressure is atmospheric pressures. This is well known assumptions what we do it and it is quite valid that whenever the fluid exits subsonically, that means the flow is subsonic level, that means the flow is less than 1 to an atmospheric.

At that point, we can assume it, the pressure is equal to the atmospheric pressure, the pressure is equal to the atmospheric pressure. Similar way, as I said it earlier also, you always choose the inlet outlet, which will be normal to the flow condition. That is what I said it earlier this the scalar product between the velocity vector and normal vector to the control surface that should have a either $\theta = 0$ or $\theta = 5^\circ$. So it is very simplified, we can solve the problems.

Otherwise, it can be done it with a scalar product of these two factors, it is not that difficult, but it is a laborious, maybe you need more time to solve the problem as compared to if you take a appropriate control volume, all the control surface. Then you can solve the problem easily. Let me repeat these things what I have talked to you, which is very important when you apply the linear momentum equation is that, the linear momentum equations what we get it is a vector equation.

It has three scalar components, X directions, Y directions, and Z directions. Many of the times when you solve our problems, you just apply one only one moment equations in one direction. You neglect other directions or that direction not necessary. But we remember the momentum equations is a vector equation. It has a three scalar directions equations we can write it. Second things which is again I am to repeat it that always you do not compete on the sign convention.

What time the momentum flux will be positive or the momentum flux will be the negative. Just look it that what will be the scalar product of the velocity vectors and normal to the control surface. Is it $\theta = 0$ or $\theta = 5^\circ$. That way it will be define you that whether you will be positive fill or the negative fill. So, do not confuse that why this sometimes we consider momentum flux is positive, sometimes momentum flux is negative. So, these because of the scalar product.

The sign of the scalar product what you have consider it, between velocity and the normal vector to the control surface. The fourth one what again I am going to repeat it any real life problems the velocity distribution is not uniform. There is always a velocity distributions as they are from the pre surface to the surface of contact. There will be a velocity distribution. What we do it, we do not consider, always you do not consider integrations of that.

Instead of that we consider distortions of velocities, compute the momentum flux correction factors for that type of flow velocity distributions and that correction factor we use and the average velocity to compute momentum flux. So momentum flux correction factor and average velocity we have used to compute momentum flux. What is going through a control surface, where you have the velocity distribution, it is not uniform. If you uniform it, velocity is the same throughout this control surface.

Beta becomes 1 and last two points, what I am talking that many of the cases what you have considered this flow exits into the atmospheric at the subsonic flow levels, that means flow Mach number is less than 1. In that case, you can consider the pressure is equal to the atmospheric pressure. And the last point already I discuss is that, we always choose the control surface such a way that it should have a normal vector and the velocity vector should be having an angle of $\theta = 0$ or $\theta = 5^{\circ}$.

That is the main tips what we need to do it before applying the linear momentum equations. We remember it, it is very easy to solve any complex problems if appropriate, linear control volume we consider and you have assumptions and all things you clearly noted. Looking that, I know it this problem what I am going to solve with these four examples, can be solved directly putting the momentum equations. But I do not follow that.

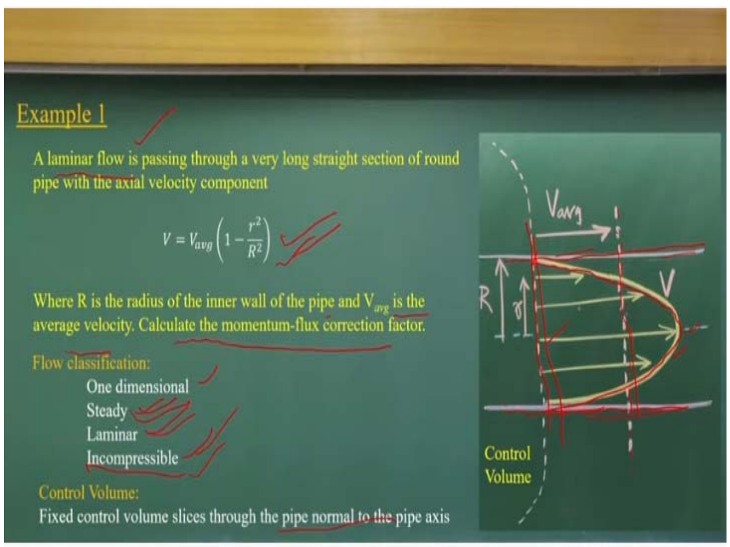
I follow very systematically the first flow classifications, second I do appropriate control volume, skies pressure diagram, force diagrams, then velocity diagrams, then apply the continuity equations, finally I use the momentum equations. So, if you look it that, whenever you get a problem, do not be excited that to apply the linear momentum equations as of you got the equation, simplify questions. Instead of that, please follow these procedures.

That first you do a classifications of fluid flow problems, what type of problems you have. Second, you try to stage the control volumes, which is appropriate control volumes and the control surface. Third, you look it, what could be the pressure distributions. Fourth what could be the velocity distributions or momentum flux correction factors. There you go for, applying the mass conservation equation and linear momentum equation.

Please follow these procedures, then if you follow these procedures, I think doing the error during the solving the problems that what will be nullified it. Otherwise, if you apply the linear momentum equations without a proper control volume, it is expected that you will do a mistakes compared to if you follow appropriate control volume, all the assumptions are clearly highlighted, then you can solve the problems and that is what you will have a chance to do the error will be nullified as compared to solve directly using linear momentum.

To demonstrate that, I am just going to solve these four problems and you look it, how I am doing the assumptions one by one, then I am applying the mass conservation equations and the linear momentum equations, not directly applying the mass conservations or linear momentum equations. Yes let us solve four examples considering the concept what I said earlier. First examples I have taken it that a linear flow is passing through a very long straight round pipe with external velocity components.

(Refer Slide Time: 32:23)



[A laminar flow is passing through a very long straight section of round pipe with the axial velocity component

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

Where R is the radius of the inner wall of the pipe and V_{avg} is the average velocity. Calculate the momentum-flux correction factor.]

So velocity components is given it, how it varies it, okay. This is the five, okay. If R is a radius of inner wall of the pipe, V_{avg} is the average velocity, then calculate the momentum flux correction factors. This is very simple problems. If you look it, you first draw the sketch. The sketch says that you have the pipe, okay and there is a velocity distribution, which is equal to r square. So that way, the velocity is maximum at this point, R that what will be the velocity becomes 0.

So, that way, you will have the velocity distribution what you would, approximately you could plot it, okay, what could be the velocity distribution. And somewhere, this average velocity will come it like this, okay. So this is the velocity distributions, the laminar flow through a long straight pipe, this is the tentative velocity distribution. Now we are going to compute it, what could be the momentum flux correction factor, if this is the velocity distribution.

Flow classification:

One dimensional

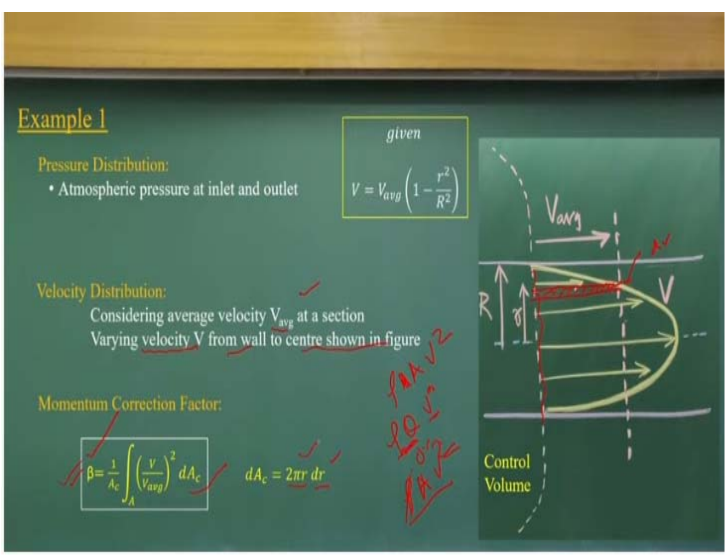
Steady

Laminar

Incompressible

No doubt, there is no time component is there, so this is a steady flow. Flow is laminar, is already highlighted and we can assume it, in this case the pipe flow may not have the Mach number of more than 0.3, the flow can be considered as incompressible flow. Here okay, will not consider this control volume, okay. We can consider as a control volume and slicing the pipe normal. Just you consider a control volume, where you just want to make it what will be the velocity distributions.

(Refer Slide Time: 34:35)



Now, we have these distributions like this and since I have the average velocity V_{avg} at a sections and the velocity V vary from wall to the centers as given in this figures. Then I am to compute the momentum flux correction factors, what is that? We need to compute the momentum flux. If I consider the velocity distribution, divide by the momentum flux. If I consider V = V average, the ratio between these two terms will give me the momentum flux correction factors, if you remember the definitions.

Pressure Distribution:

- Atmospheric pressure at inlet and outlet

Velocity Distribution:

Considering average velocity V_{avg} at a section

Varying velocity V from wall to centre shown in figure

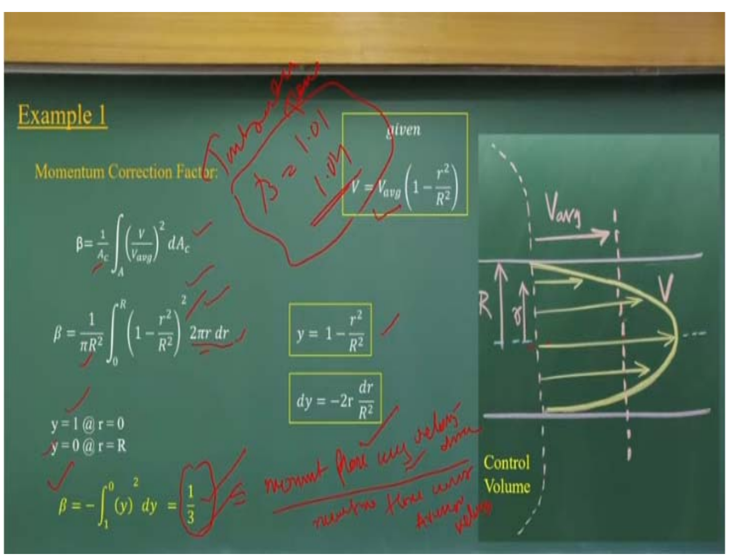
So, if you apply it, what is the momentum flux? ρAV^2 or ρQV, ρQ is my mass flux, V is the velocity, mass into velocity is the momentum flux. ρQV or ρAV^2 is a momentum flux. Now, if ρ is a constant, does not vary it, so it will be cancelled out it, then you will be V^2 by V average square by the dA_c. So please remember it, it is a very simple concept is that the momentum flux what, ρQV and if I substitute Q equal to AV, then ρAV^2.

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

$$dA_c = 2\pi r\, dr$$

That is the point what we will do it. If I have considered element area dr, the area will be the circumference into the dr, that what will the area, length into dr will be the area part.

(Refer Slide Time: 37:14)



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

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

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

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.

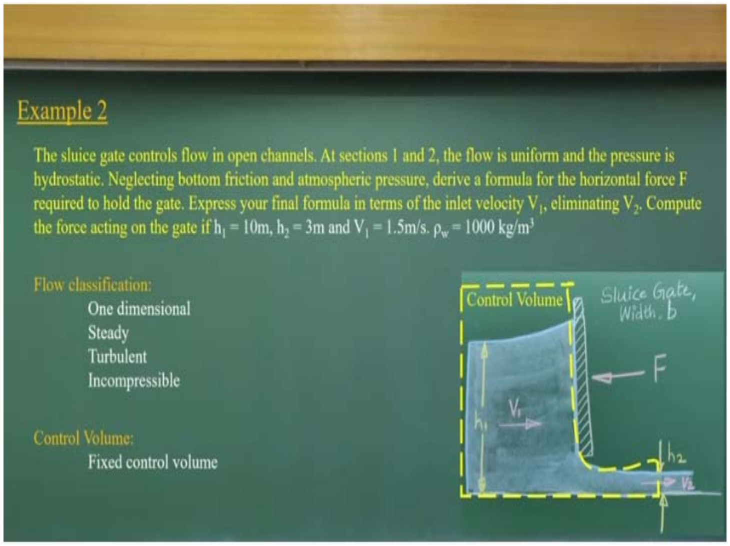
And if you substitute these values, and you will get it one by third. So, in laminar flow case whatever this velocity distributions, the beta factor is called, comes to what one by third. That means, if you computing the momentum flux using these average velocities, the actual momentum flux going through 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 $h_1 = 10$m, $h_2 = 3$m and $V_1 = 1.5$m/s. $\rho_w = 1000$ 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

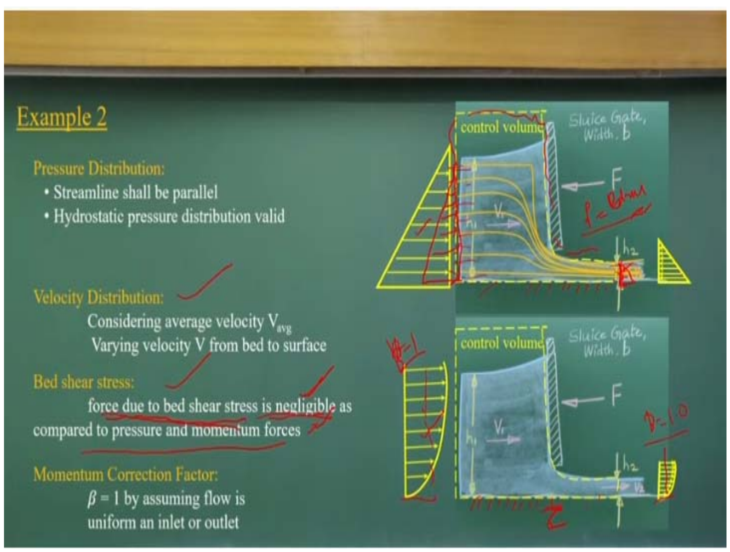
Steady

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.

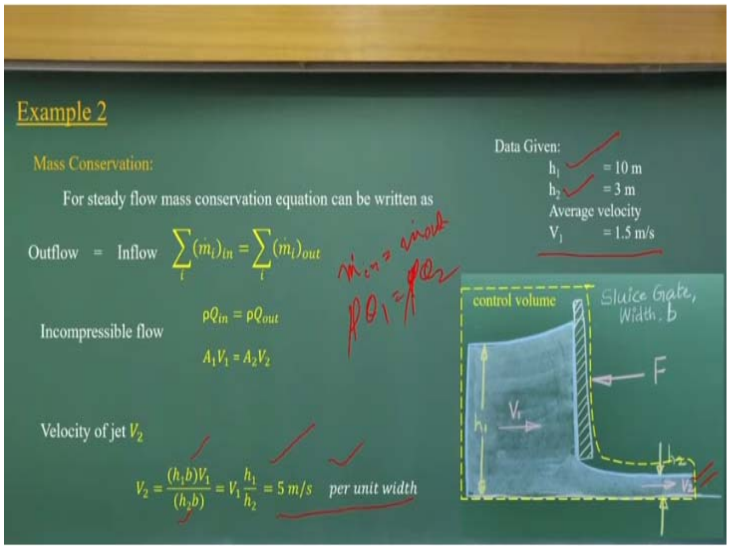
Now, you have to look the velocity distributions. As you expected that, here this velocity distribution will not be uniform, okay. The velocity distributions will be there, the zero velocity distributions at the wall and this value we get. So, in this case we consider is average velocity, okay. We do 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

$$A_1 V_1 = A_2 V_2$$

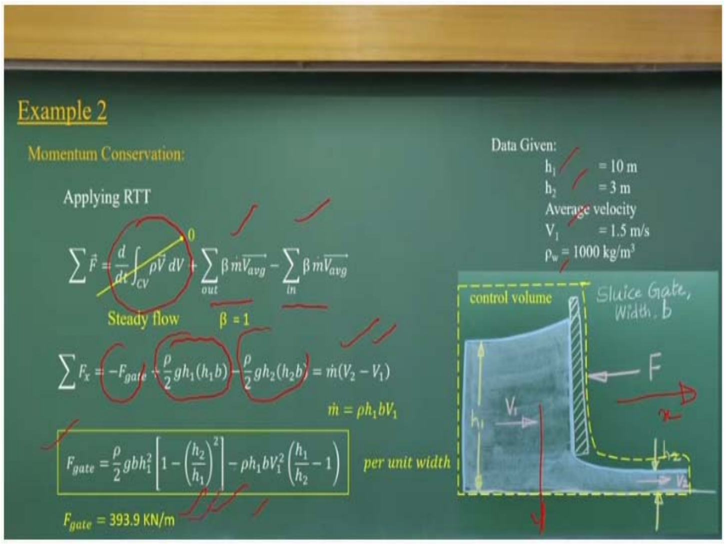
So, you know, h_1, h_2 you can consider a unit width perpendicular to the surface then the V will be cancelled out or you can V width, then you can compute the what will be the velocity. So, if I substitute, h_1 = 10 meters, h_2 = 3 meters, the average velocity what is coming 1.5 meters, as with depth 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

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

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.

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

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

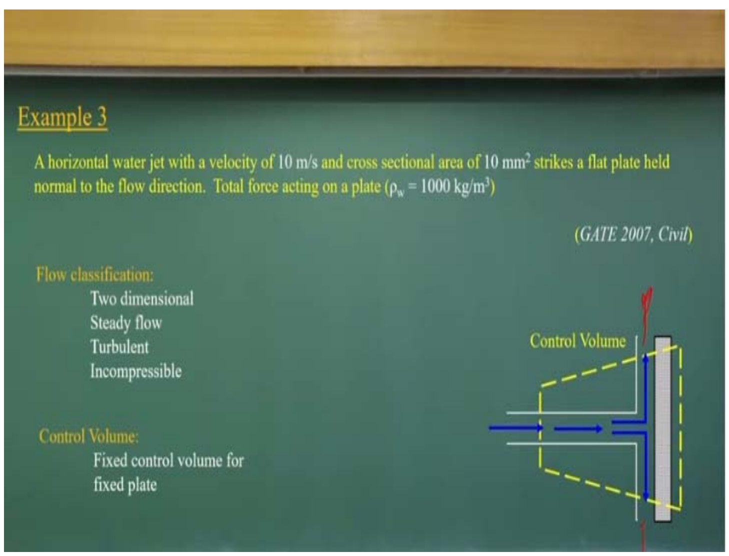
Not in the X directions, my body force is acting on the Y directions or the Z directions. So I will have a force gate is equal to, this is the pressure force component at these locations equal to rate of change of the momentum flux, the mass flux will be the same, b2 – b1, that is what will be the rate 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

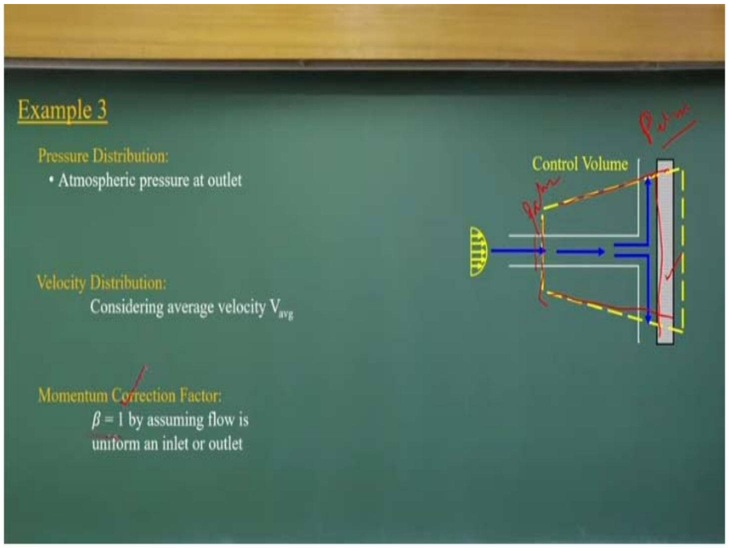
Incompressible

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.

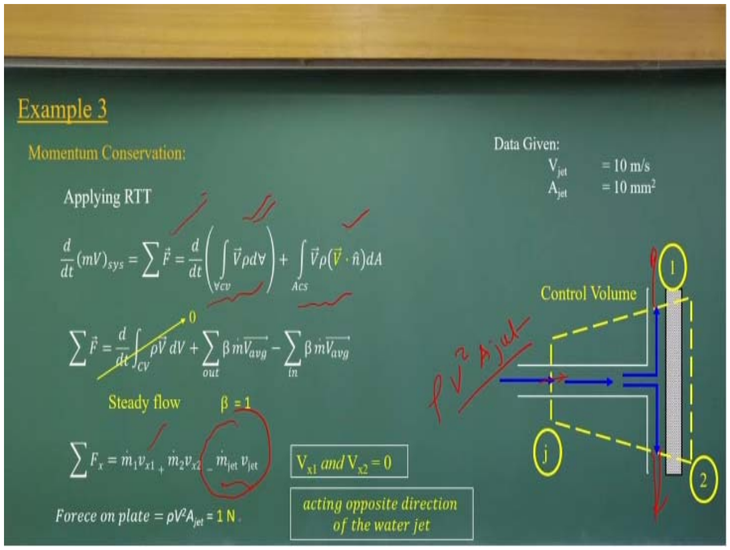
So more or less, the same amount of the flow will go from this direction and this directions. So, that way, what our jet is we gained that, we can consider a fixed control volume.

(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.

0

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

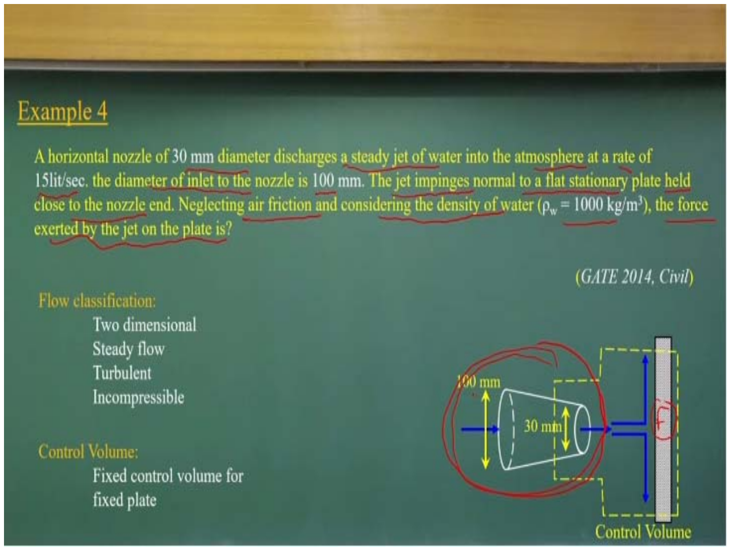
$\sum Fx = \dot{m}_1 v_{x1} + \dot{m}_2 v_{x2} - \dot{m}_{jet}\, v_{jet}$

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

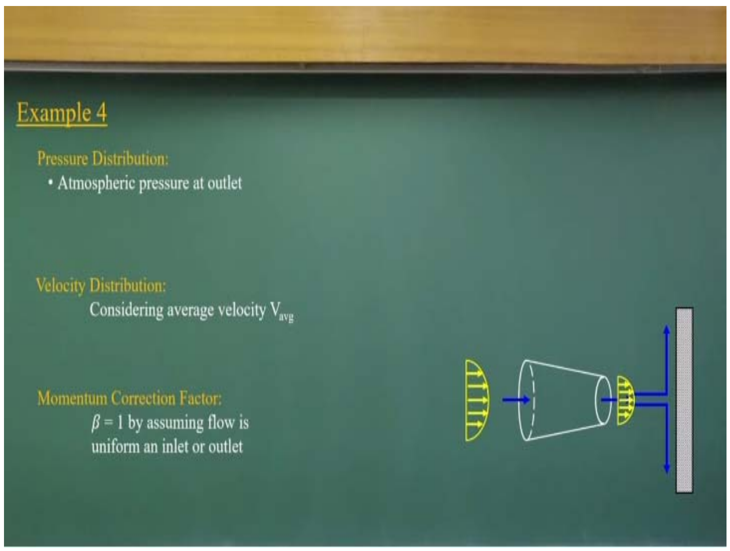
Turbulent

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.

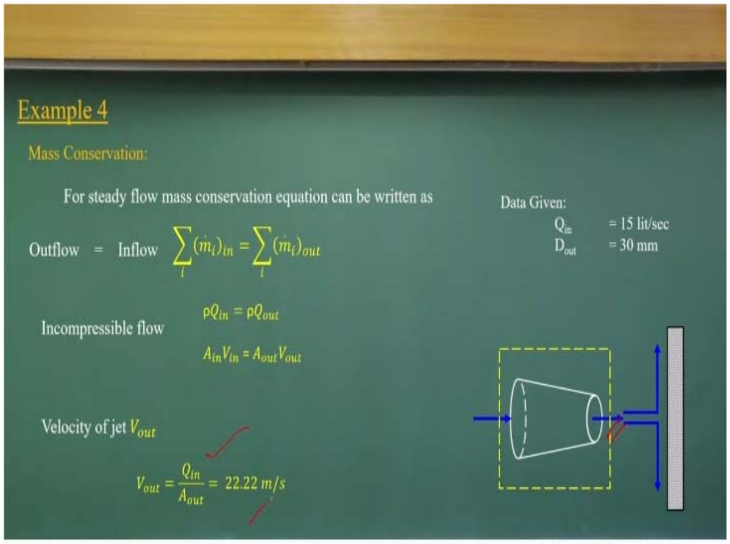
So again, the flow classification is two dimensional, steady flow, turbulent and incompressible, fixed control volume.

(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

$$\text{Outflow} \quad = \quad \text{Inflow}$$

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

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

that is why Incompressible flow

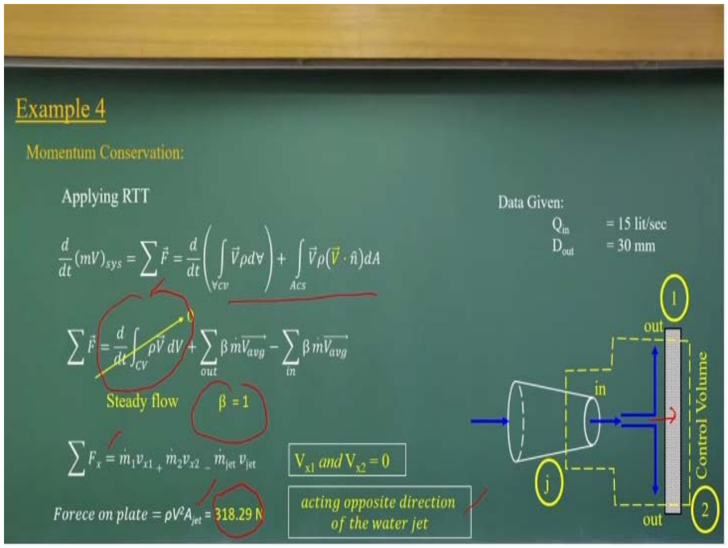
$$A_{in}V_{in} = A_{out}V_{out}$$

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

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

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

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.

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

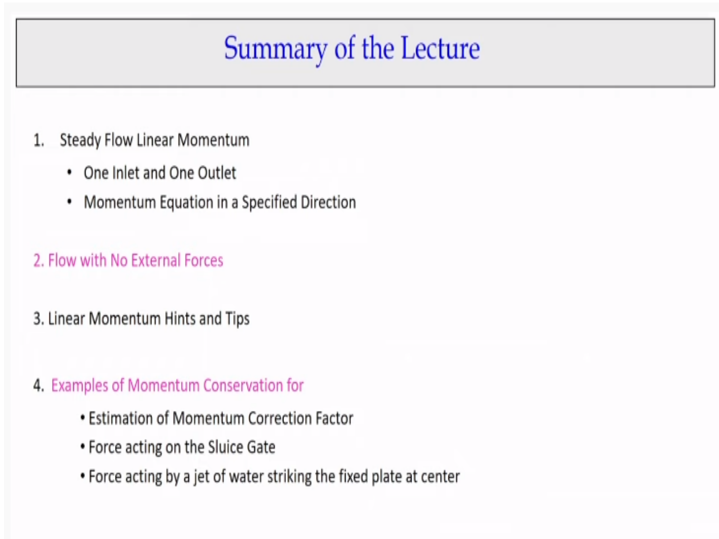
V_{x1} *and* $V_{x2} = 0$

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