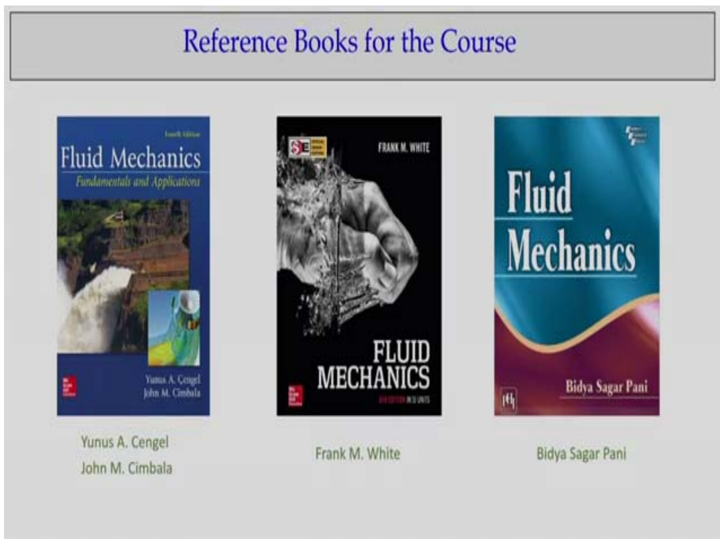
Fluid Mechanics
Prof. Subashisa Dutta
Department of Civil Engineering
Indian Institute of Technology Guwahati

Lecture No. – 09
Conservation of Momentum

Welcome all of you to fluid mechanics course. Today, I am going to deliver lecture on conservation of momentum. As you know, in the last class we discussed about conservation of mass. Also, we have solved few problems based on the conservation of mass.

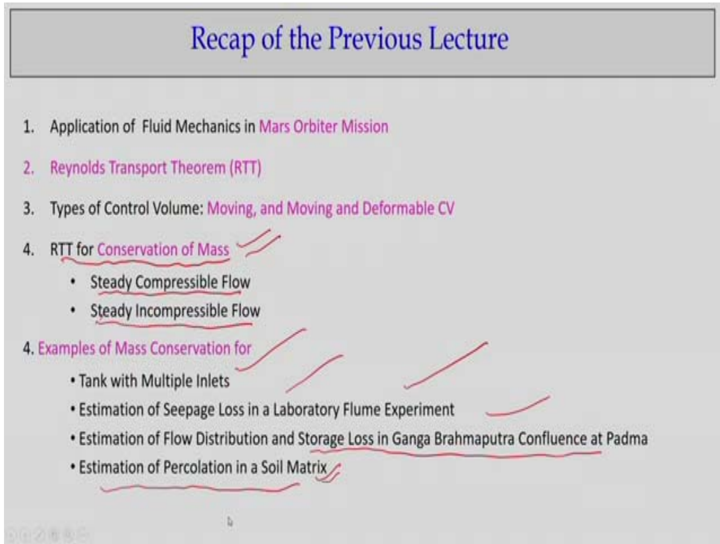
(Refer Slide Time: 00:57)



Basically, we have been following the Reynolds transport theorem as a basic concept to apply the setup of the system into physical equations to the control volume level. And, then, at the control column level, we have approximation of extensive and intensive properties and as of now we have derived mass conservation equations. Today, I am going to derive conservation of linear momentum.

Again, I can tell you, book wise, the Cengel and Cimbala, The Fluid Mechanics Fundamentals and Applications. This book has given very clearly the illustrations of this concept of Reynolds transport theorem, the conservation of mass, conservation of linear momentum, angular momentum, and the energy concept which is more descriptive type and that is why it gives enough to a student to understand this concept.

(Refer Slide Time: 02:05)



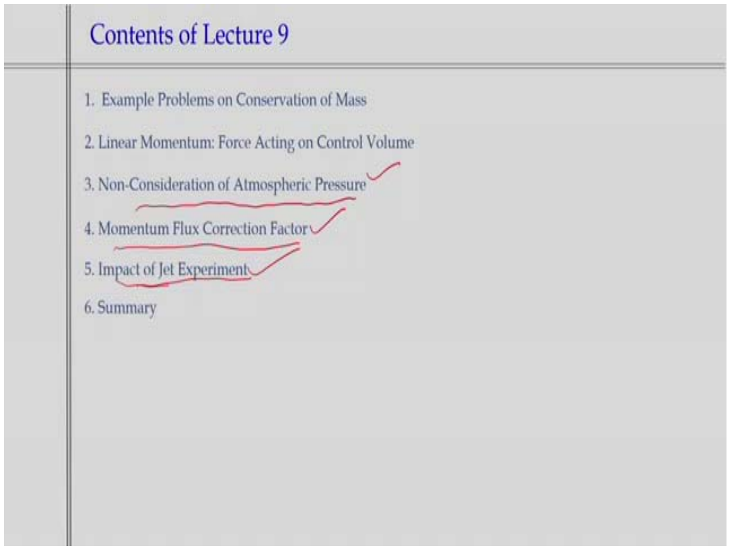
So, now let us come back to the last class what we studied. As I told you, we discussed about the Reynolds transport theorem for conservation of mass and when we apply this conservation of mass to the Reynolds transport theorem, we have two basic assumptions, that is, with respect to time is it a steady or unsteady. So, the steady we do the approximations of many fluid flow problems which are steady problems.

Then, with respect to density change or the variations of the density, we divide it, flow is compressible or incompressible. So, we can have two types of approximations, steady compressible, steady incompressible. So, when you have the steady assumptions, you can remember that the component of Reynolds transport theorem of time, differentiate components become 0 or the volume integral component part of the Reynolds transport theorem becomes 0. So, it becomes a very easy problem.

You have only this surface integral component and it is equal to 0. So, that is a very simplified case. And when the density is a constant, that means, what happens is densities comes out from the equations which makes us only the scalar product between velocity and the normal vectors, that is what is a scalar quantity. We do surface integrals with respect to area.

So, thus, the problems becomes too simple as compared to if you have compressible flow. So, when you have a steady incompressible flow, most of the case what we consider for flow devices or engineering applications, we can consider steady incompressible flow, then the problems becomes very simplified when you apply for Reynolds transport theorem. As you remember, there are starting with tanks with multiple inlets, the estimation of seepage losses in laboratory flumes. Today, I will repeat the problem, how to do these things.

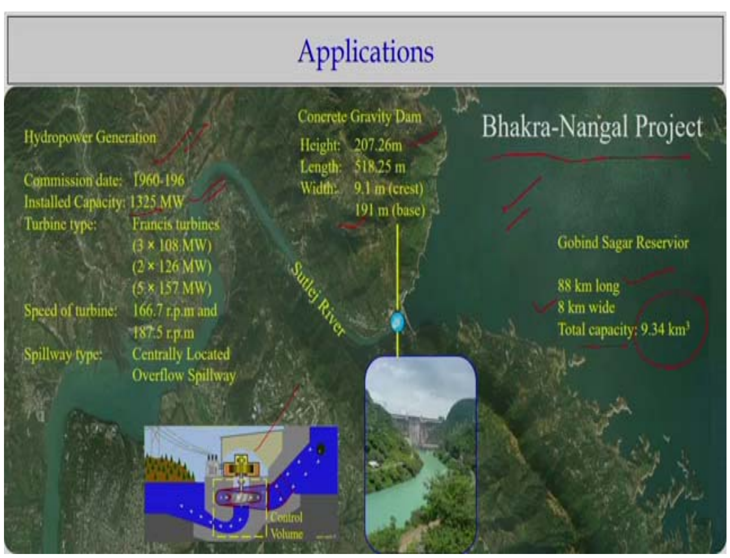
(Refer Slide Time: 04:45)



Let me come back to today's lectures, what I will cover. Again, I will give you a few examples on conservation of mass. Then, we will go to write the linear momentum equations for fixed control volumes or moving control volumes. Then, what are the simplifications that need to be done before applying linear momentum equations. That is what we will discuss in terms of non-consideration of atmospheric pressures.

Then, what is called the momentum flux correction factor, how we use it, that is what I will discuss. Then, I will show the impact of jet experiment, then, I will summarise it.

(Refer Slide Time: 05:40)



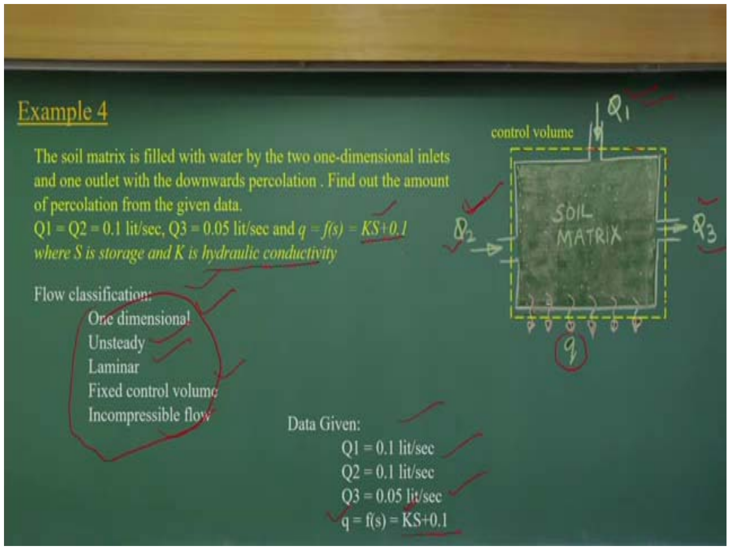
So, before going to these things, you could have heard of this hydro projects, one is one of the largest projects in our country, which is Bhakra Nangal project. If we look at this Bhakra Nangal project, it has a reservoir which is about 88 kilometers long and 8 kilometers wide. And total water storage capacity is about 9.34 kilometer cube, so, huge amount of water storage if you can see in this Google earth imagery.

The dam is located here which is a concrete dam and having a height of 207 meters approximately, and the length is 500 meters, and width varies from at the top 9 meters, as it goes down the base becomes wider and wider which will be 191 meters. So, what I am to say is that, if you look at this project which was initiated or commissioned early in 1950s and 60s, generating and installing hydro power projects about 1300 megawatt power.

So, what you are looking is basic fluid mechanics knowledge. That is what is used to design this hydropower project. So, the basic fluid mechanics what we have that is what is used way back in 1950s to design this Bhakra Nangal project which is one of the successful projects in our country. So, if you look at this way, we will take a lot of hydro power projects and we will tell how to estimate the power potentials, how to estimate what could be the turbine speed, all we can do it.

It is not a difficult task if you have knowledge of fluid mechanics. So, only the knowledge of fluid mechanics and civil engineering excellence is helping us to generate the power at the order of 1300 megawatt powers without polluting the environment. So, the hydropower projects they have the strength. Also, some disadvantages are there, but they are the projects that are implemented and those project components we can understand if we understand fluid mechanics well. That is my point for you.

(Refer Slide Time: 08:10)



For this, let us come back to the example. Last time we discussed this problem. Again I am going to repeat it just for more detailed understanding of these problems. Let us consider that there is a soil matrix, that means there are soils that are there which is having porous space, and in that soil component we have the flow. The water is coming, it is Q1, Q2, and Q3 is going out. And at the bottom, there is percolation or seepaging.

[The soil matrix is filled with water by the two one-dimensional inlets and one outlet with the downwards percolation. Find out the amount of percolation from the given data.

Q1 = Q2 = 0.1 lit/sec, Q3 = 0.05 lit/sec and $q = f(s) = KS+0.1$

where S is storage and K is hydraulic conductivity]

The water is coming out from the soil matrix. In the porous space of the soil water is there, that is what is coming out as seepage water to here. Here, this q is a function of storage within the soil matrix and the K is a constant proportional or we can call hydraulic conductivity. So, Q1 is given for this study, Q2 is given, Q3 is given. So, I have taken this is my control volume. If you look at the yellow colors it is the control volume.

If that is the control volume, before applying this conservation of mass I should classify the problem. The problem is what nature, it is one dimensional flow. The flow what we can consider across this control surface is one-dimensional.

Flow classification:

One dimensional

Unsteady

Laminar

Fixed control volume

Incompressible flow

Data Given:

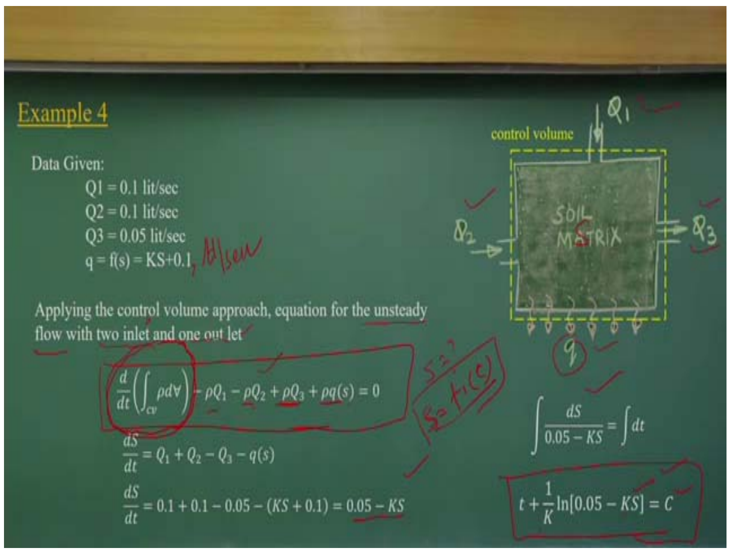
Q1 = 0.1 lit/sec

Q2 = 0.1 lit/sec

Q3 = 0.05 lit/sec

q = f(s) = KS+0.1

(Refer Slide Time: 10:24)



Now, I have to simplify the problem. I have to apply under this control volume the basic mass conservation equations. It is unsteady equation with two inlets and one outlet. That is what you can do, Q1 and Q2 are inlets, Q3 is outlet. Then, the outlet is a seepage part which is going out which will be in terms of S. If you look, when I consider this is the control volume having Q1, Q2, the inflows, and Q3 is outflow, and q is also outflow, which is a functions with respect to S, the storage.

Applying the control volume approach, equation for the unsteady flow with two inlet and one out let

$$\frac{d}{dt}\left(\int_{cv} \rho d\forall\right) - \rho Q_1 - \rho Q_2 + \rho Q_3 + \rho q(s) = 0$$

The control volume's K is 0.1 which unit will be litre per second. Then, I just apply the unsteady flow equations of convergence of mass. This is the volume integrals part if you remember it, and since it is a one-dimensional part we have the negative for the inflows and positive for the outflows. So, you can find out this Q3s and all. All are the mass flow in, mass flow out, and that is the integration. And this part is the storage, S.

$$\frac{dS}{dt} = Q_1 + Q_2 - Q_3 - q(s)$$

$$\frac{dS}{dt} = 0.1 + 0.1 - 0.05 - (KS + 0.1) = 0.05 - KS$$

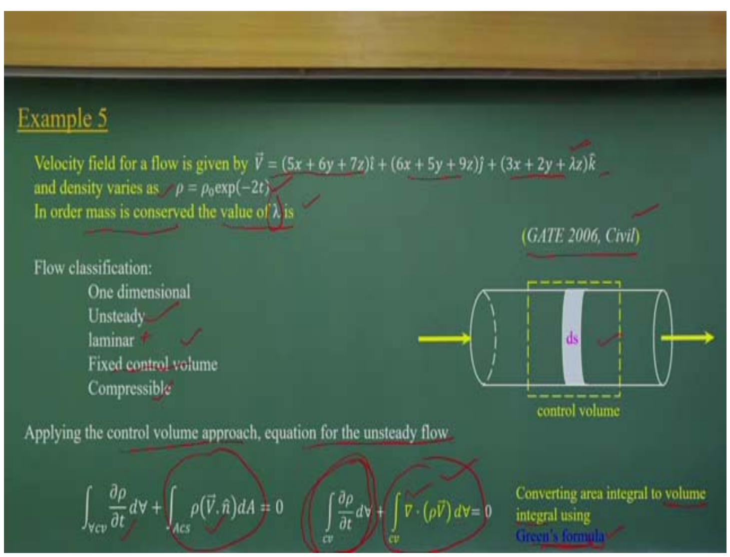
So, dS by dt, Q1 plus Q2, and this is just rearrangement of this and substituting this value we will get the dS dt is these functions. And we can integrate it and finally get a relationship between d and S as K is the constant, is equal to C. So, if you have the boundary conditions we can determine the C value, then we can know what is the function of S, how the S varies with respect to the time. That is our problem.

$$\int \frac{dS}{0.05 - KS} = \int dt$$

$$t + \frac{1}{K}\ln[0.05 - KS] = C$$

So, that way, if you look, very complex problems like this, when you have a soil matrix and porous structure and you have the flow of Q1, Q2 inflows, and outflow is there, the seepage is a function of how of water storage within the soil matrix. We can apply a simple mass conservation equation for this control volume. Then we can integrate it to get what is the function of S with respect time and that is what will give us from this. So, this is about the problem. Again, I solved it for you.

(Refer Slide Time: 13:21)



So, now, let us come to example five which is the GATE 2006 civil engineering part. In that problem the velocity field is given. If you see this, the scalar component of the velocity field in which there is a component of lambda is unknown to us, okay? Density also varies with respect to the time. That means it is unsteady problems. And the velocity fields are given.

[Velocity field for a flow is given by $\vec{V} = (5x + 6y + 7z)\hat{\imath} + (6x + 5y + 9z)\hat{\jmath} + (3x + 2y + \lambda z)\hat{k}$ and density varies as $\rho = \rho_0 \exp(-2t)$

In order mass is conserved the value of λ is

(*GATE 2006*, *Civil*)]

And here that mass is conserved, the point is that mass is conserved, then what could be the value of lambda. That is what will be different. So, that means what we will do is we will apply the mass conservation equations and once that mass conservation equation is satisfied, from that mass conservation equation we will compute what will be the lambda value. That is the problem here. So, let me classify the problem.

Flow classification:

One dimensional

Unsteady

laminar

Fixed control volume

Compressible

That means it is some sort of the volume like this, you have a dS like this. So, applying this control volume approach, equations for unsteady flow, you will have this component and this component which already we derived earlier. Let me put it this form, okay?

Applying the control volume approach, equation for the unsteady flow

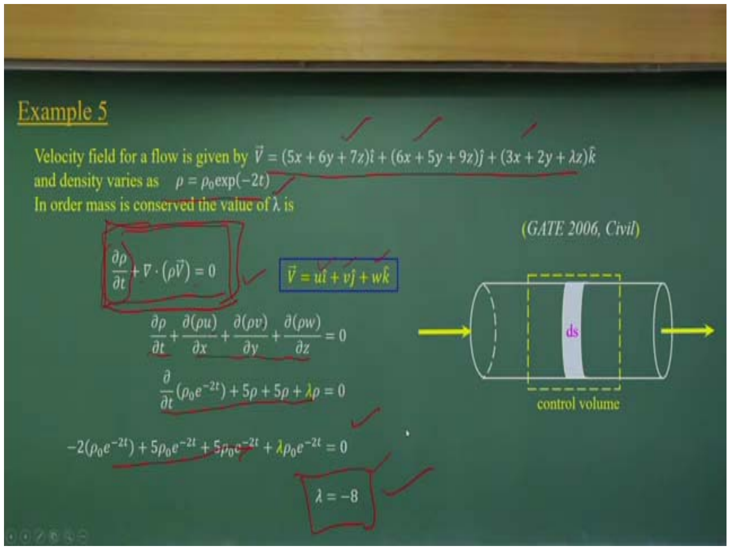
$$\int_{\forall cv} \frac{\partial \rho}{\partial t} d\forall + \int_{Acs} \rho(\vec{V}.\hat{n}) dA = 0$$

$$\int_{cv} \frac{\partial \rho}{\partial t} d\forall + \int_{cv} \nabla \cdot (\rho \vec{V})\, d\forall = 0$$

(Converting area integral to volume integral using - Green's formula)

The time derivative part of the control volumes that is what we will have this part. See, if you look this surface integrals, if I follow this Green's formula, these surface integrals can be converted to volume integrals in terms of delta operators, okay? If you remember this Green's formula, we can convert this surface integral into the control volume levels having delta dot products. That is the concept we could have known from the mathematics point of view.

(Refer Slide Time: 15:45)



Now, if I have that part and if you do the integral part, I can go out.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0$$

Then, finally, the equation becomes this because when you do integration over that, that is common, we can take it out.

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

$$\frac{\partial}{\partial t}(\rho_0 e^{-2t}) + 5\rho + 5\rho + \lambda\rho = 0$$

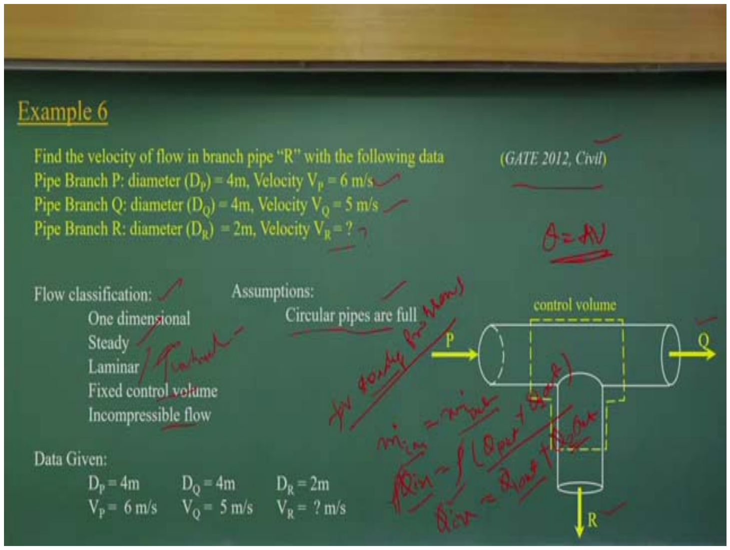
$$-2(\rho_0 e^{-2t}) + 5\rho_0 e^{-2t} + 5\rho_0 e^{-2t} + \lambda\rho_0 e^{-2t} = 0$$

$$\lambda = -8$$

Why, this u, v, w are scalar component, you can get this component as this one, okay, Finally, I get these relations here, and this relation will finally give me the lambda value which is equal to minus 8. So, this is the problem that we solved. So, basically, this equation is conservation of mass, what we have applied.

But since the velocity vectors are there which is having three dimensional velocity vectors and the density is a function of the time, so, we have applied it as bringing to this level. Then, we have solved for the lambda value.

(Refer Slide Time: 17:09)



Now, we take another example which is given in GATE 2012 civil engineering specialization. There it is a very simple problem, like what is given in this diagram. The pipe is there and there is a joint which is called a T-joint like this. P is inflow that is coming. Q is going out from this. R is going from this out. The pipes is having branching of P, Q, R. The diameters are given. The velocities V_P and V_Q are given. V_R to be estimated which is very simplified problem. You can see that this problem is one-dimensional, steady.

[Find the velocity of flow in branch pipe "R" with the following data

Pipe Branch P: diameter (D_P) = 4m, Velocity V_P = 6 m/s

Pipe Branch Q: diameter (D_Q) = 4m, Velocity V_Q = 5 m/s

Pipe Branch R: diameter (D_R) = 2m, Velocity V_R = ?]

Flow classification:

- One dimensional
- Steady
- Laminar
- Fixed control volume
- Incompressible flow

Assumptions:

- Circular pipes are full

Velocity is given. The circular pipes are full. The flow could be laminar or turbulent, okay? We do not know it. Fixed control volume and incompressible. So, this is a very simple problem. Looking at this we will just apply the control volume and try to find out what will be the mass. Inflow is coming into this and going out. Basically, if you try to remember mass in plus mass per unit time coming in should be equal to mass inflow from going out for steady problems. So, being a steady problem, what we have.

Data Given:

$D_P = 4m$

$V_P = 6$ m/s

$D_Q = 4m$

$V_Q = 5$ m/s

$D_R = 2m$

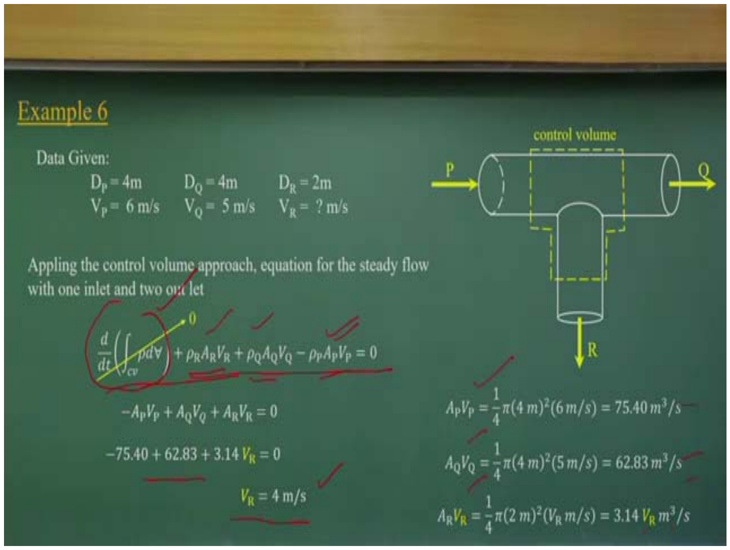
$V_R =$? m/s

The mass inflow what is coming, rate of mass inflow is what is coming in, it should be equal to rate of mass inflow going out from this control volume, that is the thing. So, in this case, because there are two outlets, sum of this two masses outflows going out from this, that is equal to mass inflow that is coming in, mass flow rate that is coming in, the mass per unit time that is coming in.

So, that way you can see that if you have ρ Q in, a very simple, ρ Q in will be rho Q1 out plus Q2 out. That is the basic concept. Since it is same density, that means you have Q in is equal to Q1 out or Q2 out to outlet. So, sum of the two volumetric discharge is equal to the inflow volumetric discharge what is going. That is very simple problem. Only, you have to compute. Since the velocity is given, so Q will be Q into V.

That is the basic concept, area into velocity is Q, average velocity is given to us, so we can compute the discharge and we just applied the Q in is equal to Q1 out plus Q2 out.

(Refer Slide Time: 20:00)



So, numerically that is what is coming. For the steady flow this becomes zero. So, you have in and out. As you know, this in will be negative and both out will be positives, and substituting these Q values for all the cases, with V_R unknown. So finally, substituting to this equations will give V_R equal to 4 m/s. So, very simple form of solving the pipe problems where you have one inlet and two outlets.

Appling the control volume approach, equation for the steady flow with one inlet and two out let

0

$$\frac{d}{dt}\left(\int_{cv} \rho d\forall\right) + \rho_R A_R V_R + \rho_Q A_Q V_Q - \rho_P A_P V_P = 0$$

$$-A_P V_P + A_Q V_Q + A_R V_R = 0$$

$$A_P V_P = \frac{1}{4}\pi(4\,m)^2(6\,m/s) = 75.40\,m^3/s$$

$$A_Q V_Q = \frac{1}{4}\pi(4\,m)^2(5\,m/s) = 62.83\,m^3/s$$

$$A_R V_R = \frac{1}{4}\pi(2\,m)^2(V_R\,m/s) = 3.14\,V_R\,m^3/s$$

As it is incompressible flow, the density is constant, so you just do volumetric flux coming in is equal to the sum of the volumetric flux going out from the control volume. That is what we commit. If you do not remember that, very simple way you remember it is that the mass influx or rate of change of mass with respect to time coming into the control volume should be equal to the rate of the mass going out from the control volume, that should be equal.

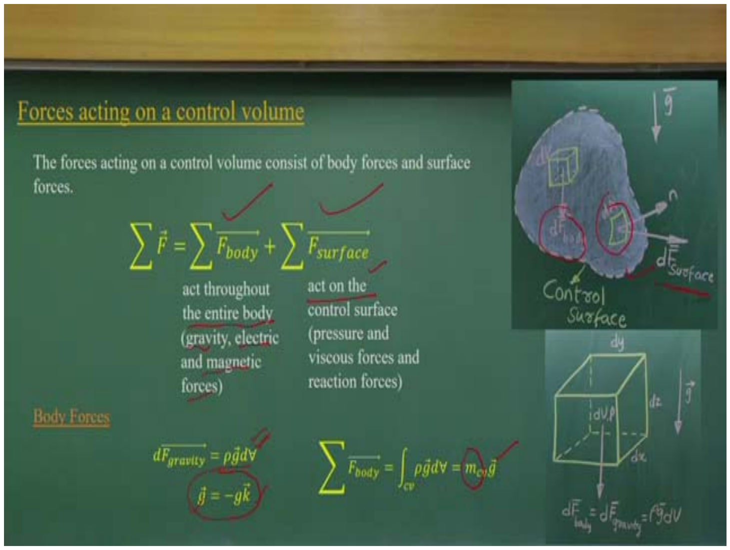
$-A_P V_P + A_Q V_Q + A_R V_R = 0$

$-75.40 + 62.83 + 3.14\, V_{\mathrm{R}} = 0$

$V_{\mathrm{R}} = 4\ \mathrm{m/s}$

Mass influx and outflux rate should be equal. That is the concept if we consider for steady flow conditions.

(Refer Slide Time: 21:25)



Now, let us come to derive the linear momentum equations, okay? So, we are going for solving these flow problems for linear momentum equations. That means we will consider the control volumes, we have considered the control volume like this. So, each control volume has the control surface. It could be a very simple tetrahedral type of structure or you can have very complex, it does not matter what could be the shapes, okay?

It can have simple shapes or it can have very complex shapes. So, if you look at that, over that surface what will happen is you will have normal vectors, let dA be the surface area, over that is the normal vector to the surface area. So, you will have the surface which will have two types forces going to act on this. One is the body force, that because of the mass of the control volume, how much of body force is giving, say, gravity point of view, or other forces we do not consider here is electrical or magnetic field point of view.

So, basically, because of the control volume mass, how much of weight you are getting, how much of body force you are getting, that is what will be the body force. And along the surface the forces that is acting that is surface force. So, you have two force components you get for a control volume. One is the body force and the other is surface force. The body force acting in the entire body, that is what is most often the gravity.

The other electric and magnetic fields are not considered for this case, but some cases we can consider it. The other is the surface force acting on the control surface. So, the forces acting on the control surface will be the pressure force, the viscous force, and any reaction forces, okay. Like a control surface is cutting through a surface, a rigid part. So, there could be a reaction force that will be there.

$$\sum \vec{F} = \sum \overrightarrow{F_{body}} + \sum \overrightarrow{F_{surface}}$$

So, that way if you look, if we take a control volume there results to be two types of forces. One is the body force which acts throughout the entire body, that depends on the mass present within the control volume. Another is that over the surface of control volume there will be the force components, those forces are due to pressure, viscous forces because of viscosities of the fluid flow systems, there will be the viscous force component, and also reaction forces or the other force component comes in.

Now, let me find out what will be the gravity force, which is a very easy thing. If I take a small element dV, I will have the weight of these small control volumes, it will be ρ, g, and dV. So, look at the unit of each component, if you can understand that. ρdV will be the mass, d V is here. Look for the volume. Mass into g is the gravity force component. Here, the gravity force component, we can consider g is a vector quantity of any direction.

$$d\overrightarrow{F_{gravity}} = \rho \vec{g} d\forall$$

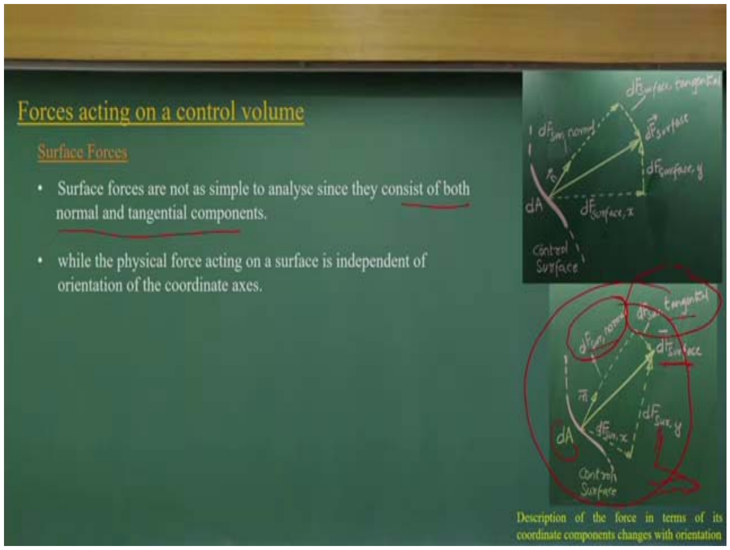
$$\vec{g} = -g\vec{k}$$

But you can align with, if y is of direction, then the K notation we can use to define the g vector component, okay? So, basically, if I consider the total control volumes, then the sum of the force or volume integrals of this component ρ g d V that will be the gravity part or indirectly this is mass of the control volume into g, g is the acceleration due to gravity vector component. And many of the times we align the z-axis and g is downward, then we use the negative, not the scalar quantity, as a vector representation.

$$\sum \overrightarrow{F_{body}} = \int_{cv} \rho \vec{g} d\forall = m_{cv}\vec{g}$$

But if consider different orientation of the control volume, then you can consider g is a vector quantity, it has a scalar component of gx, gy, gz in three respective scalar direction of x, y, and z.

(Refer Slide Time: 26:18)

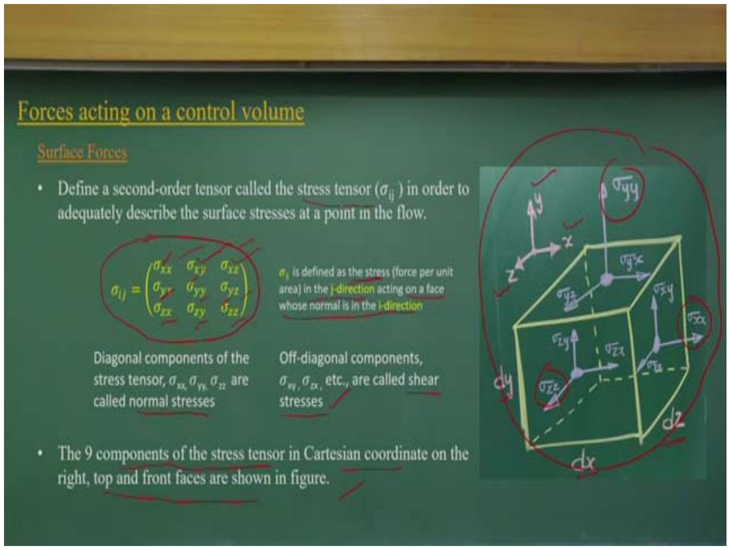


Now, let us come back to what type of force are acting. Surface forces as we discussed earlier will be there. Any surface force will have the normal component as well as the tangential component. Let us take this figure which is very interesting figure, showing to you. This is the control surface having the area of da. It is normal vector, n is this part, it is a normal vector.

So, if your force acting on this is having an angle, then this force can have two components. One is for the component for the normal, another is the tangential component, okay? So, the control surface can be considered in any orientation, okay? Over that you have a normal vector which is normal to the control surface there. So, if your force is acting at that point having a different angle, then what will happen is you will have a normal component and also the tangential component.

If you want the result as the x and y component, that is your Cartesian coordinates, that is what is different here. One is the Cartesian coordinate level resolving the force vector component to a scalar component in x and y direction. Another one is we are resolving this force component into the normal or the tangential component. That is what is illustrated here, how you can have two different components.

(Refer Slide Time: 28:13)



Now, if you come back to the surface forces, like for example, for tetrahedral structures like this where you have dx, dy, and dz components are there, and your axis is like this, x direction, y direction, and z direction. So, you can define the surface forces as a stress tensor. Stress means what, force per unit area. So, you can define as a stress tensor. So, this stress will have nine components. You could have this knowledge in solid mechanics.

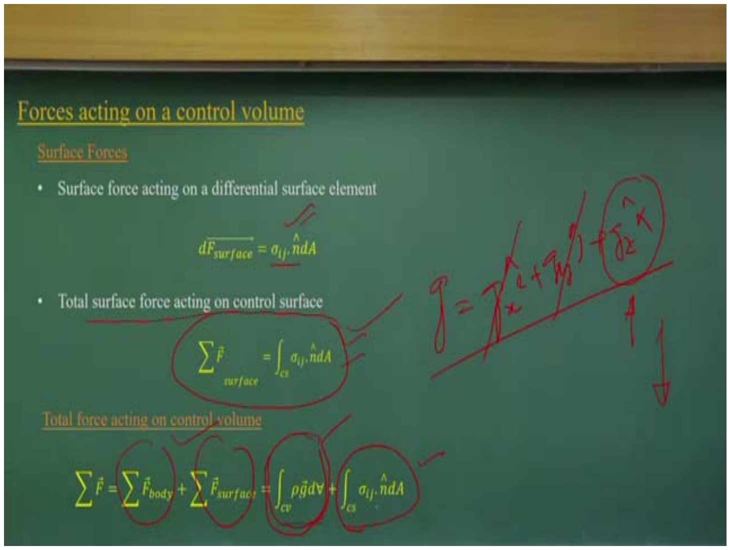
I am just repeating it. There is not much difference between solid mechanics and fluid mechanics when you consider at stress level. So, we have stress tensors in order to describe all surface force components. That is what will have nine components which will have, as you know the subscript describes that. The stress in the z direction acting on the face whose normal is eight directions. This is similar notation to what we use in solid mechanics.

$$\sigma_{ij} = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}$$

So, you have the stress tensor coordinate systems defining nine stress components. If you look at that, if you take this diagonal component which is the normal component to this surface like σ_{xx}, σ_{zz}, these are all normal components. That means these are compositions of the pressure force and the viscous force component. But the diagonal component what we have is σ_{xy}, σ_{xz}, and all, which is acting tangentially.

So, basically these are the viscous terms. So, over the surface we can define it, which is the shear stress components or the viscous stress component. So, these nine component of stress in Cartesian coordinate is defined in this surface. So, we can solve the problems considering the surface force defined as stress tensors and defining as normal stress and the shear stress component. The normal stress is a composition of pressure and viscous stresses, whereas shear stresses is only the viscous stress that we get.

(Refer Slide Time: 30:55)



Surface force acting on a differential surface element

$$\overrightarrow{dF_{surface}} = \sigma_{ij}.\hat{n}dA$$

Now, if I have the stress component there and I have the normal vectors, if I resolve the force components, I will have the scalar product between the stress tensor and the n vectors, that is how we do it. And for the total surface area we do surface integrals to compute it, okay? Please do not be more worried about how we are having a scalar product of stress tensors and normal factors which will be coming to be again a second order vector components.

You try to get the mathematical point of view of that or different literatures nowadays available, you can understand the physics behind that, how mathematically we represent this stress tensors and the dot product or the scalar product of the stress and the normal vectors or any vector quantity. So, if it is that, you can integrate it to get all the stress components. So, total force acting on the control volume will have the body force component and surface force component.

Total surface force acting on control surface

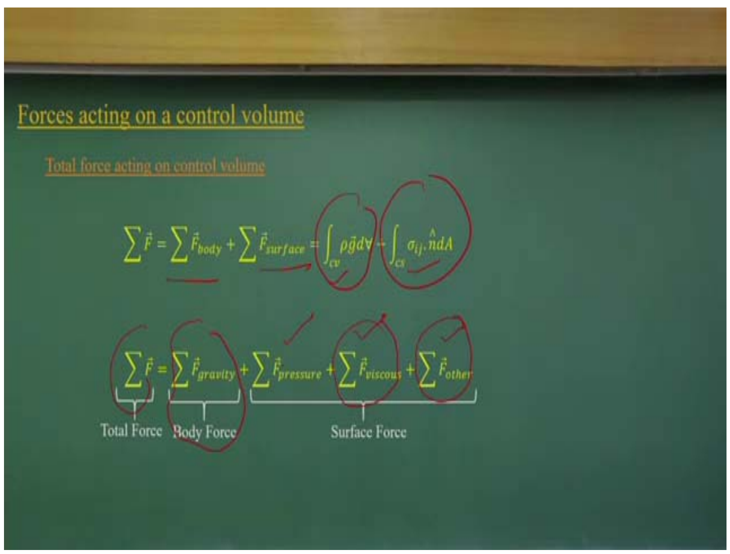
$$\sum \vec{F}_{surface} = \int_{cs} \sigma_{ij}.\hat{n}dA$$

The body force component will have volume integrals of rho g dV, g is the vector quantity as we consider the g, acceleration due to gravity can have a vector commodity with three scalar components of g_x, g_y, g_z. That means what I am defining is $g = g_x I + g_y j + g_z k$. So, you can define like this. But many of the times we make alignments in such a way that only this k direction or negative direction and k is in upward direction.

We define the g that becomes a scalar quantity acting downwards. That is what happens. But if you are solving the problem where you do not know the direction of g vectors which is equal to $g_x I + g_y j + g_z k$, then you need to do the volume integrals to solve the problems and this is the surface integrals over this control surface to get what is the force acting on the surface.
Total force acting on control volume

$$\sum \vec{F} = \sum \vec{F}_{body} + \sum \vec{F}_{surface} = \int_{cv} \rho \vec{g} d\forall + \int_{cs} \sigma_{ij}.\hat{n} dA$$

(Refer Slide Time: 33:22)



Then, we will go for simplification of this one. If you look at these things, it looks very complicated. We cannot apply simple example problems that we encounter as a civil engineer or mechanical engineers. What is the total force acting on the surface will be the body force and the surface force and this two integrals will tell me, one is volume integrals and other is surface integrals.

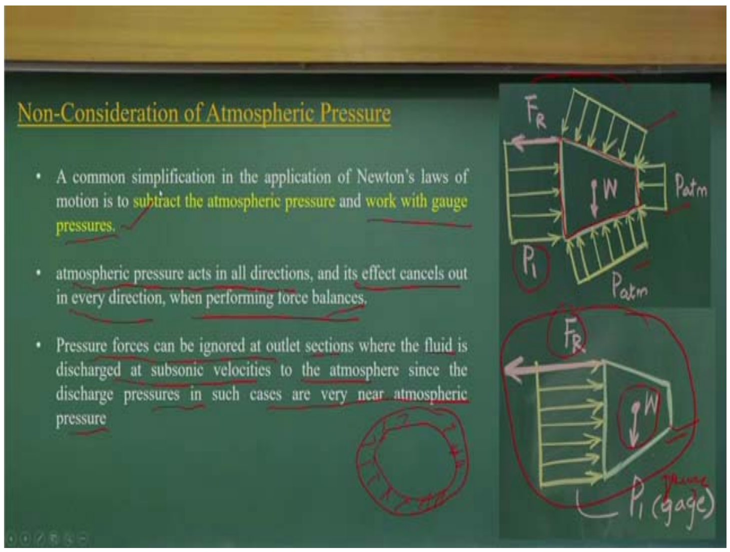
$$\sum \vec{F} = \sum \vec{F}_{body} + \sum \vec{F}_{surface} = \int_{cv} \rho \vec{g} d\forall + \int_{cs} \sigma_{ij}.\hat{n} dA$$

Now, if I divide the force components, that means, the total force will be one component from this body force, it is gravity force component. The surface force component we can resolve it into the force due to the pressure, force due to viscosity, force due to the other reactions. That means the reaction component of force. See, if I resolve this force component and if we can have a simplification like the cases where the viscous force is not dominated, then we can make it 0, or other forces not dominated we can make it 0.

$$\sum \vec{F} = \sum \vec{F}_{gravity} + \sum \vec{F}_{pressure} + \sum \vec{F}_{viscous} + \sum \vec{F}_{other}$$

That way, since we have resolved it or we separated the force components due to the pressure, due to the viscous, and separately for different types of problems we can simplify these equations and focus on the force components only due to the pressures or due to viscous or due to other components.

(Refer Slide Time: 34:54)



Now, let us consider very important component before applying this linear momentum equations. When you apply a linear momentum equations to a control volume, like I have the control surface like this, if you look at this control surface, in these three phases we have the pressure is atmospheric pressure. Here I have the absolute pressure, that means atmospheric pressure plus the gauge pressure.

If I look at any control volume I consider, the control surface always will have pressure equal to the atmospheric pressure. That means, if you can understand it, if I take a control volume, everywhere I will have atmospheric pressure, then the absolute pressure from some locations. So, if I consider the atmospheric pressure is acting throughout this control surface and do surface integral, if the pressure is having the direction, and if I do a surface integrals over this, it will be cancelled out and becomes 0.

So, considering that what we generally do it we nullify the atmospheric pressure component, because as you know when we integrate it the surface integrals of the atmospheric pressure for any control volume that becomes 0. So, we consider only the gauge pressure when you define the pressure diagrams for a control volume. For example, in this case, the atmospheric pressure components are given and this direction this pressure is equal to P_1 is the absolute pressure.

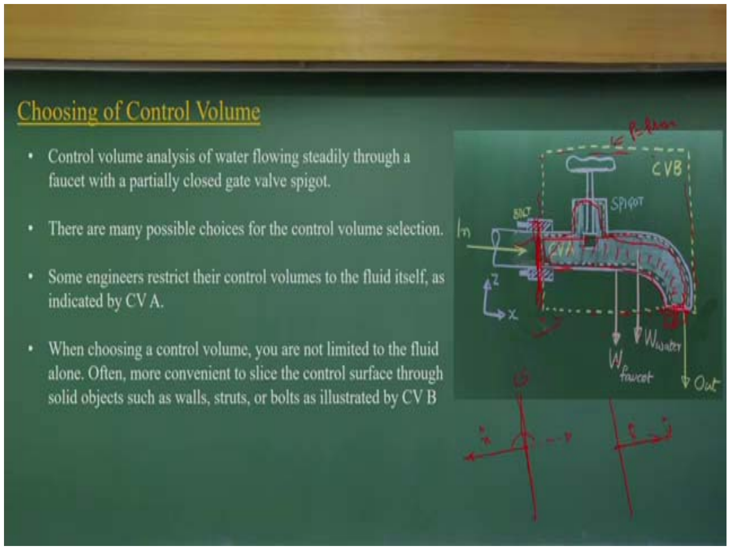
So, finally what we do is that we do not consider this pressure distribution as I said earlier. So, this part will be the gauge pressure, the difference between the absolute pressure and the atmospheric pressure. Then, we have the two force components working, one is the body force component and the other is the other reactions components. So, finally we use this control volume, a simplified control volume nullifying the atmospheric pressure distribution over the control surface.

We consider this control volume and the pressure diagram to solve the problems. That is, the atmospheric pressure acts in all the directions. Its effect is cancelled out in every direction when performing the force balancing, that is what I am saying, when you do surface integrals of this constant, pressure distribution in a close control volume, that is supposed to be 0 as it happens here. All these direction angles cancel out each other.

So, the pressure forces can be ignored at the outlet where the fluid is discharged at subsonic velocity to the atmosphere. Another assumption which is quite valid is that if you have subsonic flow, almost all times in civil engineering problems we get the subsonic flow, sonic or supersonic flow. In that case, the pressure force we generally we neglect in that state, as the discharge pressure in such cases is very near to the atmospheric pressure.

So, if you look and measure the pressure at that location when you have an outlet discharge, then that becomes the atmospheric pressure. So, we neglect that atmospheric pressure component if you have free outlet discharge. The point what I am trying to tell you is that we work with gauge pressure when you have defined the simplifications of control volume concept.

(Refer Slide Time: 39:05)



Now, another point is how to choose control volume, because that is what the art is, like you do free body diagrams in solving solid mechanics problems. Similar way, drawing the appropriate control volume is an art. That is what you have to learn by solving many problems using the control volume concept. Like, for example, what I need to do if do take this problem, okay?

So, that means there is water coming and there is a bolt holding the water outlet point and there is the spigot, and we have water coming out. I can have a control volume like as given here CV control volume, which is the wall surface touching only this water part if I consider the control volume. We do not know what is the stress acting over this surface. That is unknown to us.

Also, we do not know what is the pressure distribution at this point. So, when you draw this type of control volume, where over this control surface we do not know it, we cannot solve the problem. Instead of that, if I take a control volume like this, it is control volume B, if I look at this control volume, very clearly I know inflow, I know the weight component, one is water weight component and another is weight component of this tap or this spigot part.

Then, I also know this direction and q out from this. So, if I consider this control volume and over this P will be the P atmosphere. So, this problem becomes simple. If I just want to know how much of force is acting because of this flow orientation, so I can compute the force components to find out how much force is acting on this bolt. So, the point is that it is engineering skill or art to be developed by the students how to use the control volume concept. How to define the control volume for a flow system so that you can easily solve it.

And another thing what we do is that when you define the control volume, the control surface and the velocity should align to that. That means what I am telling you is that if my control surface is like this, if I have a normal vector to that, this is the normal vector, so V should have either 180 degrees in the same directions making the n vector and the v vector. If you do that, your scalar product becomes easy to do.

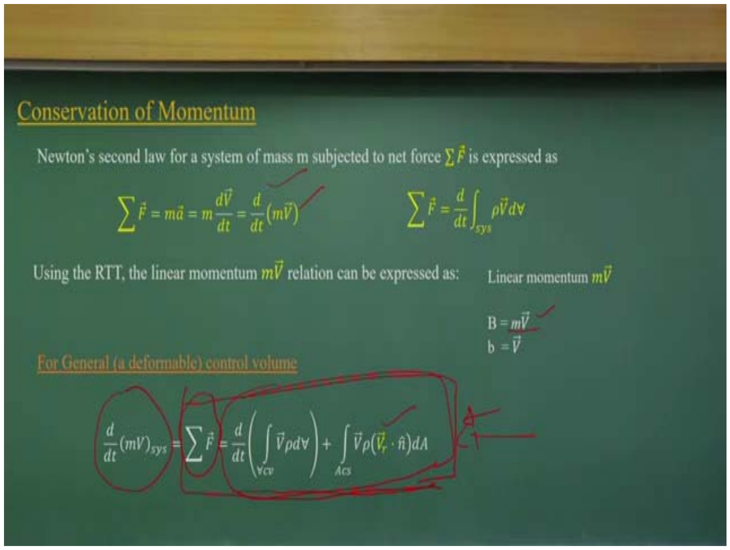
If $\theta = 0^\circ$, $\cos\theta = 1$,

if $\theta = 180^\circ$, $\cos\theta = -1$.

So, that way, it simplified this vector product. What you use in this thing is that the velocity and the normal vector they should have either 0° or 180°. That is what my point is. We can consider the control surface which will be with respect to velocity. We can take any shape, but when you take arbitrary shapes if your V and n is not having 0 or 180, finally you end up doing a scalar product, do surface integrals to solve the problem which is more time consuming and laborious, but the results will be the same.

So, what you have to look is how to define the control volume and the control surface in such a way that the target of your problems you have to solve it. That is art. That is what the art I will discuss in today's lecture. We will talk about the different types of control volume to use it. Next class onwards I will flow with how we should use appropriate control volume to solve the problems.

(Refer Slide Time: 43:27)



Now, coming back to applying the Reynolds transport theorem we have to write the linear momentum equations. At this systems level force is equal to mass into acceleration. That means, at the systems level force will be mass into acceleration as you know from basic solid mechanics. So, when I derive it at the systems level, the momentum flux, that is B, is my momentum flux. That means that should equal to net force acting on that.

$$\sum \vec{F} = m\vec{a} = m\frac{d\vec{V}}{dt} = \frac{d}{dt}(m\vec{V})$$

$$\sum \vec{F} = \frac{d}{dt}\int_{sys} \rho\vec{V}d\forall$$

That is what is the system. And at the control volume levels, as B is equal to mass into momentum flux, the v, the intensive property becomes velocity vectors. If I apply with this, I will get this equation. So, this is the general equation for a control volume. We can have a V here. For moving control volumes we can use the relative velocity component and V is the velocity vector and V r is the relative velocity vector.

Using the RTT, the linear momentum $m\vec{V}$ relation can be expressed as:

Linear momentum $m\vec{V}$

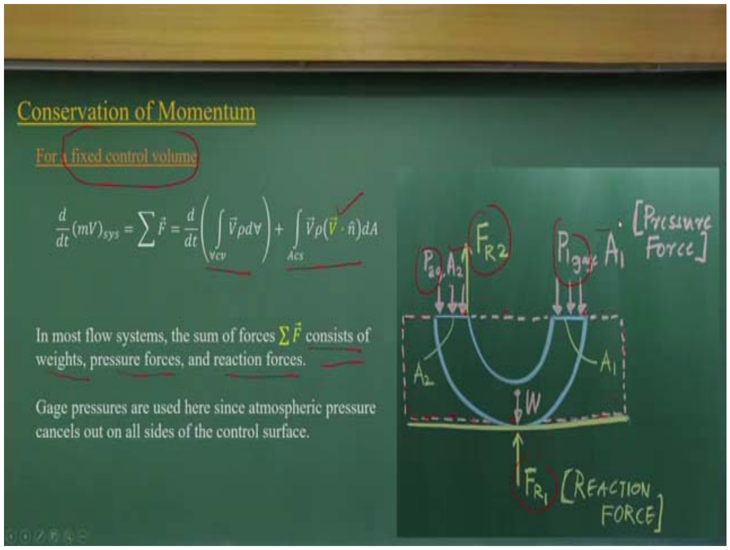
B = $m\vec{V}$

b = $\vec{V}$

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

So, this is the basic equation which is linear momentum equation in Reynolds transport theorem point of view.

(Refer Slide Time: 44:55)



This equation will be simplified to solve the problems because, as of now, this problem we cannot solve because it has surface integrals and it is also having volume integrals. So, let us consider I have a fixed control volume, that means control volume is fixed. So, your V r becomes V in the case of this. So, you have volume integrals and surface integrals like this.

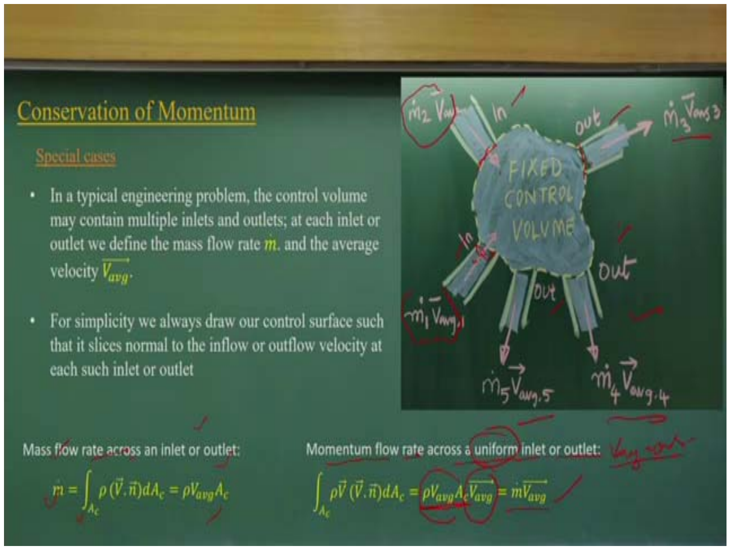
$$\frac{d}{dt}(mV)_{sys} = \sum \vec{F} = \frac{d}{dt}\left(\int_{\forall cv} \vec{V}\rho d\forall\right) + \int_{Acs} \vec{V}\rho(\vec{V}\cdot\hat{n})dA$$

This is my control volume. The flow is coming from this.

Flow is also coming from this. Both the sides flow is coming in. And there the pressure is acting on P_1 and P_2 and A_1 and A_2 and there is a weight and there is reaction force acting on that. This is a A_1 and A_2 and this is pressure diagram. If it is that, this force can have, as I said earlier, we will have body force component which is the weight, the pressure forces, and the reaction forces at this point where we are getting the reaction forces.

Also another reaction force is attached to this. Here there are two contact points, at this and this. So, we can use gauge pressure. You need not do the integration over this. You can use a gauge at this two points to find out the pressure force component due to the flow.

(Refer Slide Time: 46:18)



Now let us consider special cases. Like, many of the times, any industry if you look, there are network of pipes and connected to the tanks and all. Similar way, there will be a series of inflows, a series of outflow. If you have that type of concept, like, this is a fixed control volume, you have a series of inflows like 1 and 2 are the inflows, and 3, 4, and 5 are the outflows.

So, through this control surface if you look, we are talking about not only the mass flux coming into this control volume but we are talking about momentum flux. The momentum flux will be the mass flux, mass per unit time into the velocity, that is what will be the momentum flux. So, through this control volume we know this momentum flux. Through this control volume, what is the momentum flux coming into?

Through this control volume what is the momentum flux going out, and going out from this case, and this case, okay? So, basically, if you look, we can compute the momentum flux coming into this control volume and also going out. So, we have multiple inlets and outlet, then you can find out how much of net momentum flux is there in this control volume, just like we did for the mass flux in mass conservation equations.

Here, we are talking about net momentum flux passing through this control volume which will be equal to the net force acting on this control volume. We know force is equal to mass into accelerations. He same concept we are considering here as a change of the momentum flux within this control volume, that will be equal to the net force acting on this control volume.

So, now, if I consider velocity distributions are there, velocity variations are there, then, you know this mass flow rate, across this inlet and outlet we can have surface integrals to find out the mass flux. So,

Mass flow rate across an inlet or outlet:

$$\dot{m} = \int_{A_c} \rho\, (\vec{V}.\vec{n}) dA_c = \rho V_{avg} A_c$$

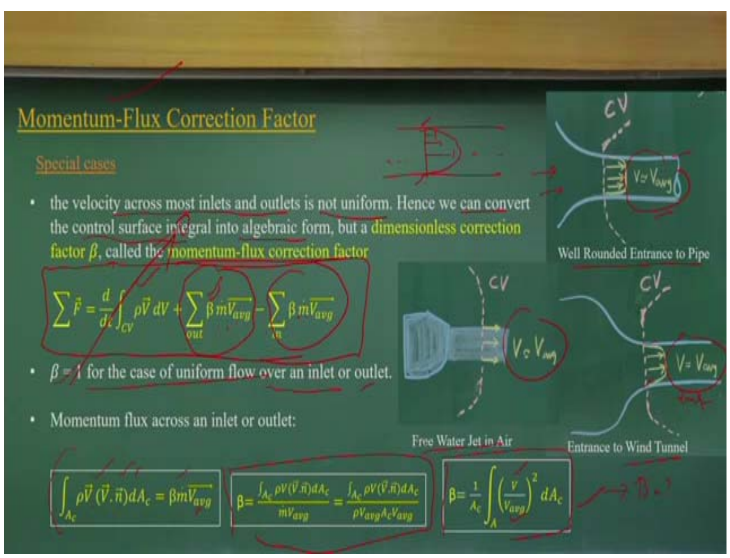
Similar way, we can find out momentum flow rate, momentum flux, if you assume it is a uniform inlet. That means your velocity is not changing. Momentum flow rate across a uniform inlet or outlet:

$$\int_{A_c} \rho\vec{V}\, (\vec{V}.\vec{n}) dA_c = \rho V_{avg} A_c \overrightarrow{V_{avg}} = \dot{m}\overrightarrow{V_{avg}}$$

V average is constant or the velocity variation is not there. That is what you call the uniform velocity. That condition does not vary. But some of the cases we can simplify that, okay? So, in that case, if in a surface velocity does not vary, so we can define it as mass flux, $\rho V_{avg} A_c$, that will be the mass flux, V average with the velocity, that will give the momentum flux.

So, here what we have considered is a uniform distributions of velocity. That is real fluid flow condition, we will have the correction factor for that.

(Refer Slide Time: 49:40)



So, if you look it that way, now we can write the momentum equations in different forms considering the velocity distribution. That is the reason we introduce correction factors. That means, as we know it, the velocity distribution is not uniform for real fluid flow problems, computing this momentum flux using the average velocity, then what could be the correction factor for that.

That is what is called the momentum flux correction factor. Let me repeat that thing. You know for real life fluid flow systems, like for example, I have three examples here, okay? One of the examples is that there is well-rounded entrance to the pipe. The flow is coming from this, very well rounded, so you will see the velocity distributions will be there. At the wall it will be 0, then velocity distribution will be 0.

Similar way, you have a larger pipe section, wind tunnel section, then it is coming as test sections like this. This is the test section, okay, wind is coming through that. So, you know, over this control surface the velocity will be 0 here, then distributions, then coming back to that, or you have free water jet coming into air. So, you will have this. Most of the times what we have is absolute, where the fluid is turbulent.

No doubt you will have 0 velocity at the wall locations. The pipes or the test section is stationary at the rest conditions. The velocity distribution is more or less uniform except at the wall location. So, we can use average velocity concept for that. The V can be approximated as average velocity for all these cases. If not, let us have the pipe flow, the laminar pipe flow, so as you know you, velocity distribution will be 0 at the pipe contact location, the maximum velocity will be at the center.

So, there will be the parabolic distribution of velocity. In that case, as the velocity is not uniform, you need to have correction factors for computing the momentum flux. If we are computing momentum flux using average velocity concept, that is what is the correction factor. So, you try to understand it. So, if you are computing the momentum flux based on the average, then you should have a correction factor for that, okay?

Let us have this, the momentum-flux correction factor which will be the dimensional correction factor of beta where the velocity in most of the inlets are not uniform, we can convert this control surface integral to algebraic form. That is what it is here, okay? Because V average computations are easy and you know this mass flux in and out for each control volume, inlet and outlets, so you can compute the momentum flux.

And since you are using the average velocity to compute the momentum flux, you also consider the velocity distribution in terms of correction factors, that is what the β value is in terms of correction. So, finally, your surface integral part and the volume integral part will come like this. If it is a steady problem, this becomes 0. Again, you have a very simplified case.

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

Sum of the force acting on that will be equal to the net momentum flux passing through this control surface, that is what will come, okay? So, let us have this for uniform flow. So, β will be equal to 1 and this thing if I do the integration of the velocities and the scalar products of V and n over a surface Ac, and that is what I am representing in average. So, β will come to be this part or with the simplification β will come this way.

If I know the velocity distribution, I know the average velocity. If I do this integration, I know this surface area, then I can compute what will be the beta value for a velocity distribution coming into a cross section. Like, I have a pipe of laminar flow, a pipe of turbulent flow, or it is an open channel flow it is connecting, but each case we know approximate velocity distribution. That means, for each case we know what is the β value.

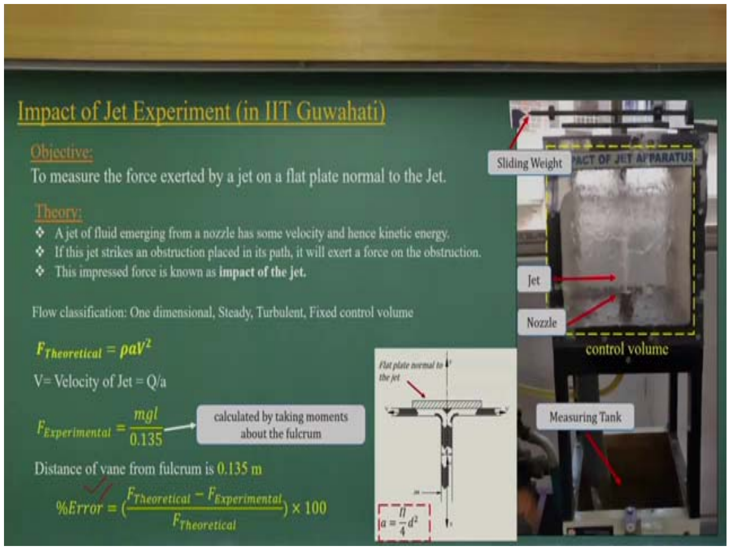
$$\int_{A_c} \rho\vec{V}\,(\vec{V}.\vec{n})dA_c = \beta\dot{m}\overrightarrow{V_{avg}}$$

$$\beta = \frac{\int_{A_c} \rho V\,(\vec{V}.\vec{n})dA_c}{\dot{m}V_{avg}} = \frac{\int_{A_c} \rho V\,(\vec{V}.\vec{n})dA_c}{\rho V_{avg} A_c V_{avg}}$$

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

So, if I know the β value, then we need do the integrations again. We use that beta value to convert from average velocity data to the momentum flux. So, that is the advantage to use the momentum-flux correction factor.

(Refer Slide Time: 55:01)



Now, before concluding today's lecture, let us see the figures, what is there, very simple experiment is impact of jet experiment. If you look at this jet, the water jet part is here and it is balanced by the weight here. So, we can know what is the velocity of impact if happening and that is how much of weight we are counterbalancing it. So, if you know it, theoretically, you know this is the control volume, you have water jet coming in and that water jet is going in these two directions.

If you look at this photograph and this simplified conceptual diagram, you can find out that. So, theoretically,

$$F_{Theoretical} = \rho a V^2$$

V= Velocity of Jet = Q/a

It is a momentum flux. ρ and V square will be the momentum flux that is impacting on that because in this direction the output momentum flux is 0. So, what is influx that is converted to the force component and you have experimental, then, definitely there is a deviation from the experiment and theoretical because any systems like this jet apparatus, it has some degree of loss.

$$F_{Experimental} = \frac{mgl}{0.135}$$

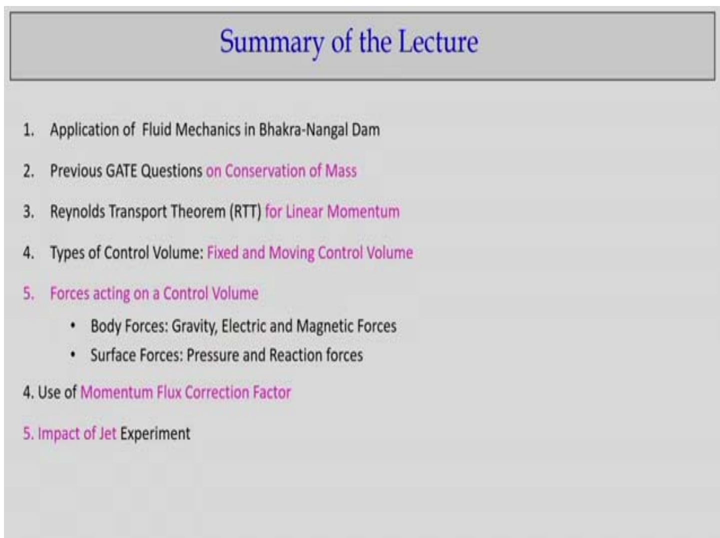
Distance of vane from fulcrum is 0.135 m

$$\%Error = (\frac{F_{Theoretical} - F_{Experimental}}{F_{Theoretical}}) \times 100$$

So, we cannot have exactly whatever the theoretical value. We will have deviations from that. That is why we compute the error. The deviations between the theoretical and experimental divided by theoretical, that is what gives us the error. So, this type of simple apparatus will be there in any engineering colleges, so you can see that how the force is acting because of jet impacting on that plate.

That is the condition. This is very simplified case. Here impact is normal to the plate. You can have many numerical examples where the plate can be inclined or the flow jet can have an incline and different conditions, we will also discuss in the next class.

(Refer Slide Time: 57:15)



So, with this let me summarise today's class. We started with very interesting Bhakra and Nangal dam project. If you are that interested, you just get more data available, but I can say that because of that dam project we have changed the irrigation, hydropower generation, and water resource management in Himachal Pradesh and part of Uttar Pradesh and all. So, all because of the knowledge of fluid mechanics way back in 1950s and 1960s, that is how that is possible.

Now, we have more advanced way to understand the fluid mechanics. As I told earlier, we can solve many, many challenging problems apart from the standard problems. And we also discussed about the Reynolds transport theorem for linear momentum equations. The problems I have not solved, in the next class I will solve the problems and try to know how to know to use the control volume appropriately so that we can solve the problem with less timing and in proper way.

That is my point. So, we can solve problems or the exercise from any of the reference books. With this, let me conclude this lecture. Thank you.