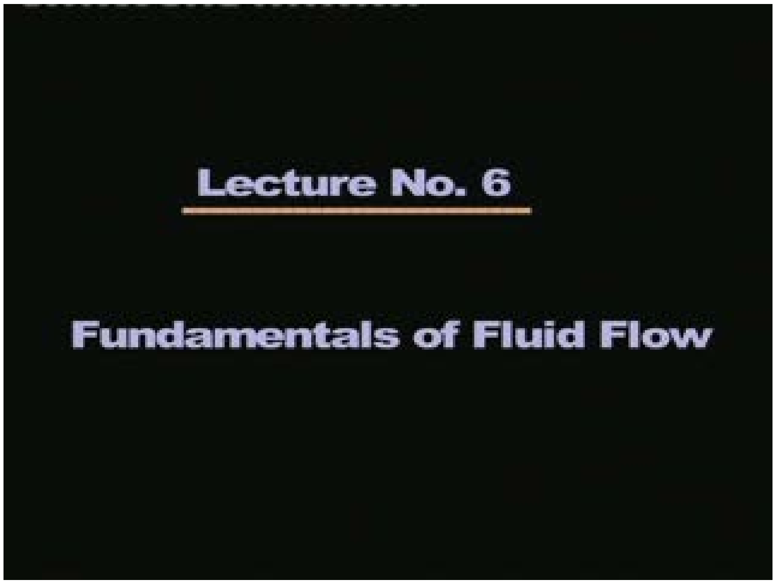
# Refrigeration and Air conditioning
**Prof. M. Ramgopal**
**Department of Mechanical Engineering**
**Indian Institute of Technology, Kharagpur**
**Lecture No. # 6**
## Fundamentals of Fluid flow

(Refer Slide Time: 00:00:45 min)



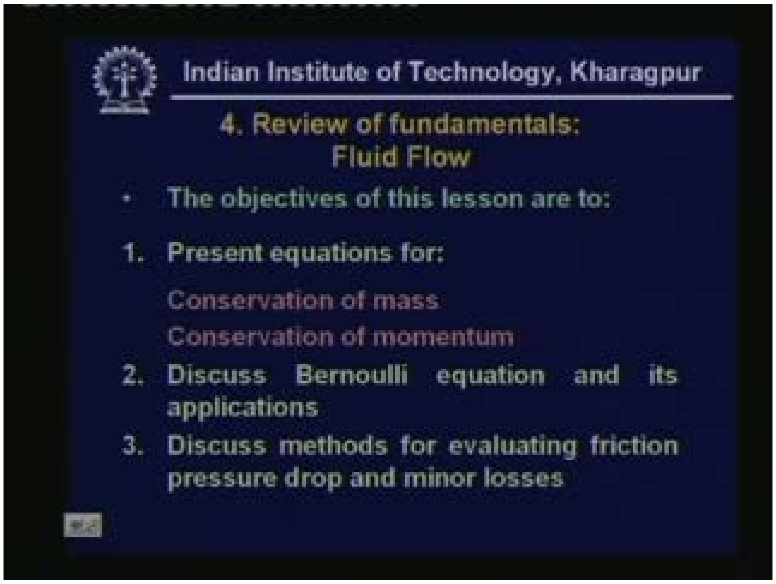
Lecture No. 6

Fundamentals of Fluid Flow

Welcome in the last two lectures. We reviewed fundamentals of thermodynamics.

So now let us review fundamentals of fluid flow in this particular lecture.

(Refer Slide Time: 00:00:59 min)



Indian Institute of Technology, Kharagpur

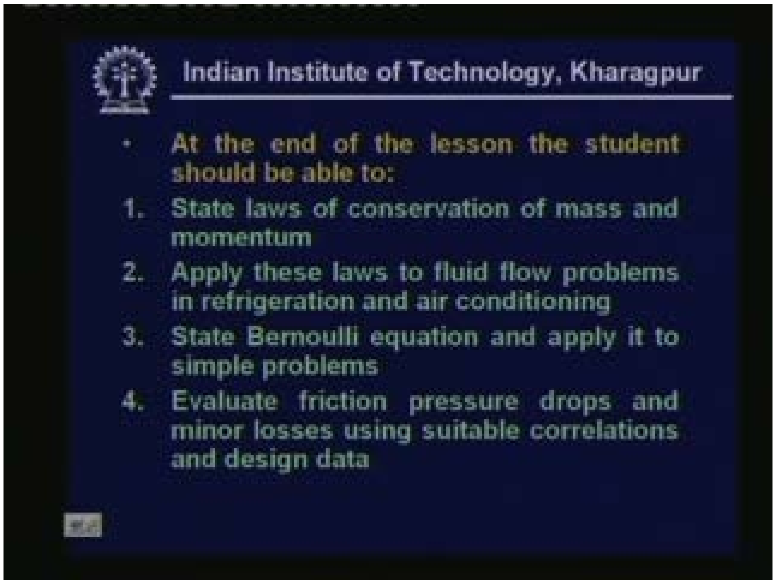
4. Review of fundamentals: Fluid Flow

- The objectives of this lesson are to:

1. Present equations for:
   Conservation of mass
   Conservation of momentum
2. Discuss Bernoulli equation and its applications
3. Discuss methods for evaluating friction pressure drop and minor losses

So the objectives of this lesson are to present equations for conservation of mass, conservation of momentum and discuss Bernoulli equation and its applications and discuss methods for evaluating friction pressure drop and minor losses.

(Refer Slide Time: 00:01:17 min)



Indian Institute of Technology, Kharagpur

- At the end of the lesson the student should be able to:

1. State laws of conservation of mass and momentum
2. Apply these laws to fluid flow problems in refrigeration and air conditioning
3. State Bernoulli equation and apply it to simple problems
4. Evaluate friction pressure drops and minor losses using suitable correlations and design data

At the end of this lesson you should be able to state laws of conservation of mass and momentum. Apply these laws to fluid flow problems in refrigeration and air conditioning and state Bernoulli equation and apply it to simple problems and finally evaluate friction pressure drops and minor losses using suitable correlations and design data.

(Refer Slide Time: 00:01:40 min)

Indian Institute of Technology, Kharagpur

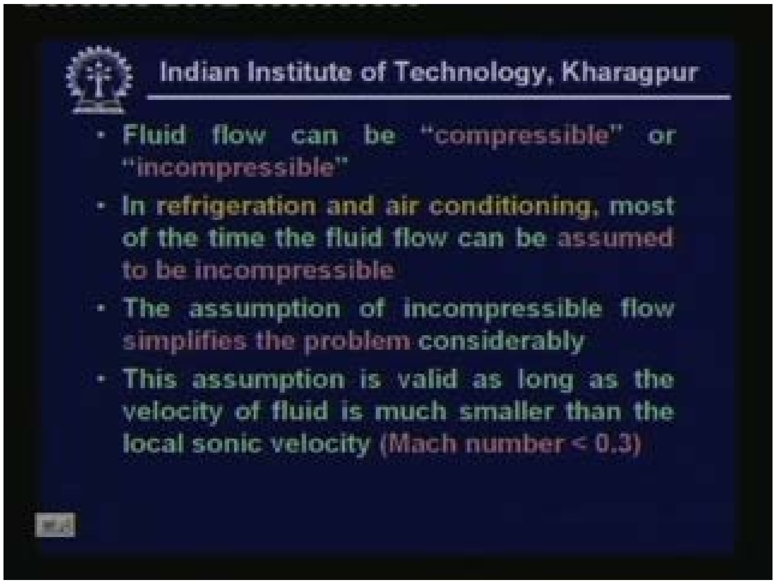
Fluid flow

- In refrigeration and air-conditioning systems various fluids such as air, water and refrigerants flow through pipes and ducts.
- The flow of these fluids is subjected to certain fundamental laws
- The subject of "Fluid Mechanics" deals with these aspects.

Now why do we have to study about fluid flow in refrigeration and air conditioning? As you have probably seen in a in a typical refrigeration system or air conditioning system we will be handling a large number of fluids for example refrigerants air water etcetera. These fluids will be flowing from one point to the other that means fluid flow involved in all these systems. Now these fluid flows are subjected to certain fundamental laws of fluid mechanics. So in order to understand or in order to design refrigeration and air conditioning system one must know the fundamentals of fluid mechanics or fluid flow. Okay.

So that is what is mentioned here in refrigeration and air conditioning systems. Various fluids such as air water and refrigerants flow through pipes and ducts and the flow of these fluids is subjected to certain fundamental laws and the subject of fluid mechanics deals with the subjects with these aspects probably we have studied the fluid mechanics. So it is not possible to review the entire subject here but we will be presenting the basic basics of these things relevant to refrigeration and air conditioning.

(Refer Slide Time: 00:02:47 min)



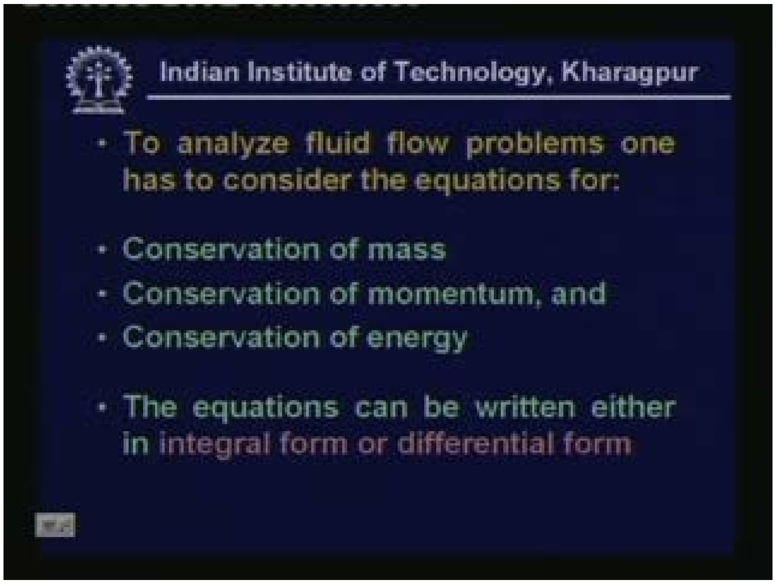
Indian Institute of Technology, Kharagpur

- Fluid flow can be "compressible" or "incompressible"
- In refrigeration and air conditioning, most of the time the fluid flow can be assumed to be incompressible
- The assumption of incompressible flow simplifies the problem considerably
- This assumption is valid as long as the velocity of fluid is much smaller than the local sonic velocity (Mach number < 0.3)

Now fluid flow in general can be compressible or incompressible. What do we mean by compressible flow? A compressible flow means the density of the fluid varies along the flow direction. That means basically fluid density varies as the fluid flows. Such flows are known as compressible flows and in incompressible flow. The density remains constant it does not vary in most of the refrigeration and air conditioning applications. The fluid flows are can be assumed to be compressible and why do we assume this because this will simplify the problem considerably.

So most of the times we assume the flow to be incompressible and the resulting mathematics will be simpler. However you cannot just like that apply the assumption of incompressible flow everywhere. You have to see its validity the assumption that the flow incompressible is valid as long as the velocity of fluid is much smaller than the local sonic velocity. That means the velocity of the fluid should be smaller than this velocity of sound or in other words the Mach number should be less than point three. And as you know the Mach number is the ratio of velocity of fluid divided by the sonic velocity. So when the Mach number is less than point three you can assume the flow to be incompressible and apply the suitable laws.

(Refer Slide Time: 00:04:07 min)

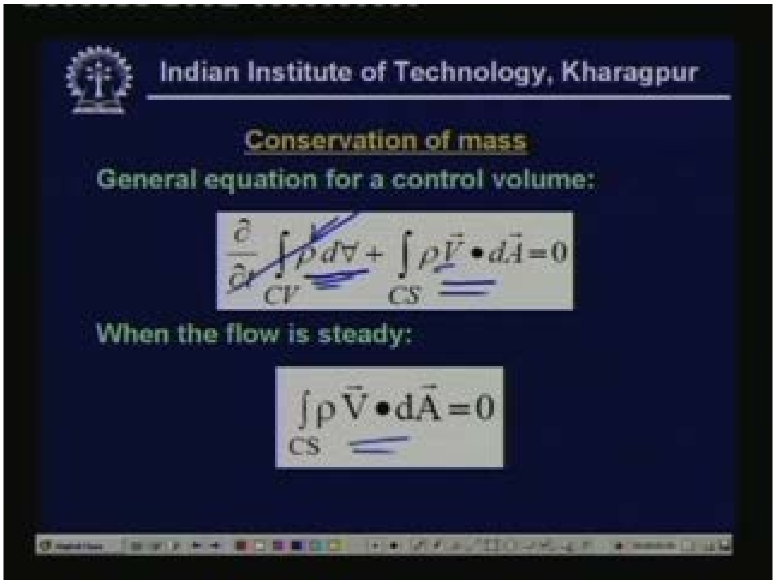


Indian Institute of Technology, Kharagpur

- To analyze fluid flow problems one has to consider the equations for:
- Conservation of mass
- Conservation of momentum, and
- Conservation of energy
- The equations can be written either in integral form or differential form

Now to analyze fluid flow problems one has to consider three conservative equations. They are equation for conservation of mass equation for conservation of momentum and equation for conservation of energy. A conservation of energy is nothing but the first law of thermodynamics which we have discussed in the review on thermodynamics. So in this pre present lecture we will be discussing conservation of mass and conservation of momentum. Now depending upon the approach you can write these equations either in integral form or in differential form. That means if you are taking control volume and apply these conservative laws you get integral form of equations or if you take a differential element inside the control volume and apply the conservative laws you get the equation governing equations in differential form. In the present lesson I will just give you only the integral forms of conservation of mass and conservation of momentum.

(Refer Slide Time: 00:05:02 min)



Indian Institute of Technology, Kharagpur

Conservation of mass

General equation for a control volume:

$$\cancel{\frac{\partial}{\partial t}\int_{CV} \rho\, d\forall} + \int_{CS} \rho \vec{V} \bullet d\vec{A} = 0$$

When the flow is steady:

$$\int_{CS} \rho \vec{V} \bullet d\vec{A} = 0$$

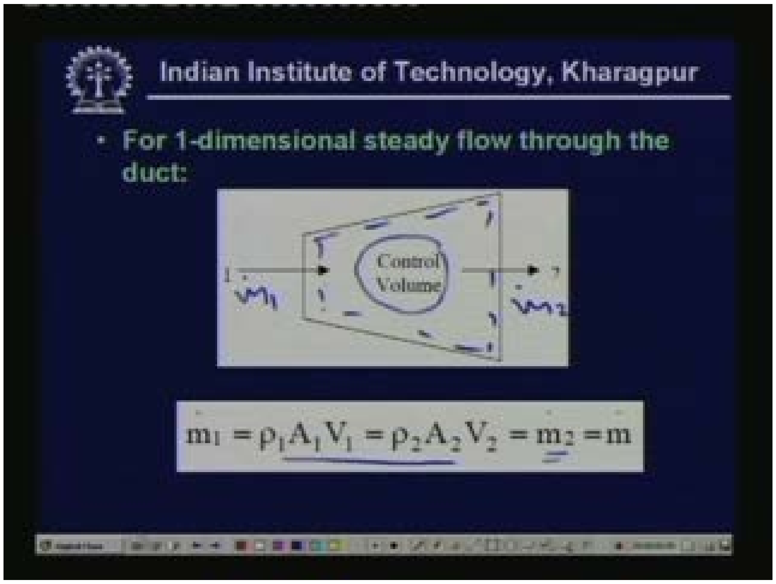
Now this equation shows the conservation of mass. As the name implies conservation of mass in simple terms is that mass can be neither created nor destroyed just like energy you cannot create mass nor you can destroy mass. So if you apply this conservation of mass to a control volume then you get a equation of this form as you can see in the in this equation you have two terms.

The first term here is, what is happening, Okay. So the first term here I am sorry is the rate at which the mass of the control volume is varying with time. That means time rate of change of mass of the control volume C V stands here for control volume.

And the second term is nothing but the net mass flux out of the control volume. So the conservation of mass when applied to a control volume gives you two terms, two integral terms since we are applying integral form. So this first term accounts for the rate at which the mass of the control volume is varying with time and the second term accounts for the net mass flux out of the control volume. In this equation rho is the density, t is the time and V is the velocity vector with reference to the control volume. Okay. And when the flow is steady that means when the flow does not vary with time this term will not be there. So this equation boils down to conservation of mass for a steady flow situation. So these equations are also known as continuity equations.

So basically for steady flows we will be having this kind of an equation.

(Refer Slide Time: 00:07:19 min)



Indian Institute of Technology, Kharagpur

- For 1-dimensional steady flow through the duct:

$$\dot{m}_1 = \underline{\rho_1 A_1 V_1 = \rho_2 A_2 V_2} = \underline{\dot{m}_2} = \dot{m}$$

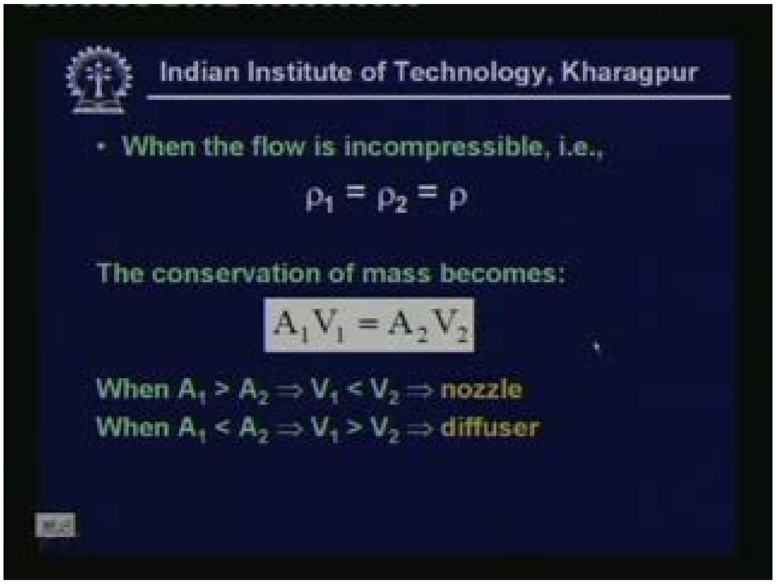
Now if you apply this conservation of mass to a one dimensional steady flow through a duct let us say that we take an arbitrarily shaped duct and let us say that this is the control volume.

And let the flow be steady and flow be one dimensional since the flow is steady no mass can be accumulated within the control volume. That means the mass inside the control volume is constant. That means whatever mass is entering that must leave the control volume. Okay.

That is what is written here the rate at which mass is entering into the control volume is equal to the rate at which the mass is leaving the control volume. And the rate at which mass is entering the control volume can be written as a product of density rho one cross section area A one at the inlet and velocity V one at the inlet.

Similarly at the outlet you can write the mass flow rate out of the control volume as rho two into A two into V two. Also this is the simple one dimensional steady flow equation when you apply it to any duct now if the density is constant.

(Refer Slide Time: 00:08:20 min)



Indian Institute of Technology, Kharagpur

- When the flow is incompressible, i.e.,

$$\rho_1 = \rho_2 = \rho$$

The conservation of mass becomes:

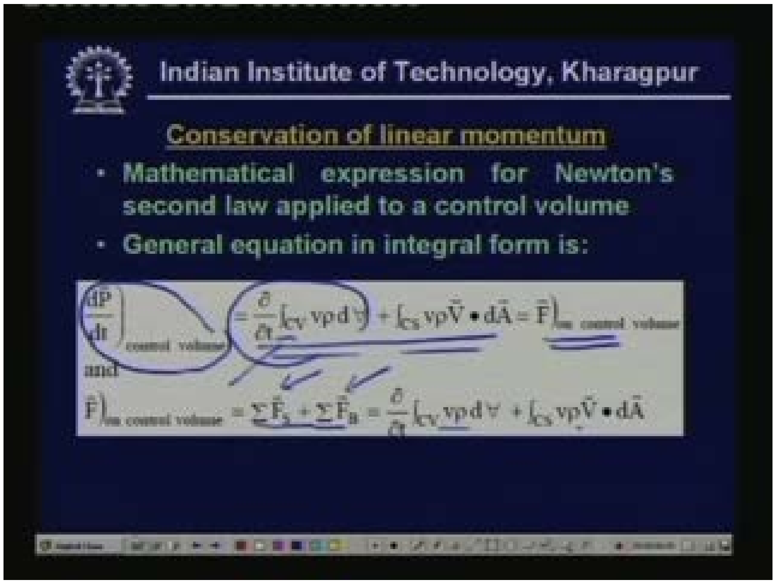
$$\boxed{A_1V_1 = A_2V_2}$$

When $A_1 > A_2 \Rightarrow V_1 < V_2 \Rightarrow$ nozzle
When $A_1 < A_2 \Rightarrow V_1 > V_2 \Rightarrow$ diffuser

That means when rho one is rho two is rho then you get a conservation of mass. If conservation of mass in a very simple form as A one V one is equal to A two V two. Okay. Right so far steady incompressible flow for a one dimensional situation we get the conservation of mass equation as A one V one is equal to A two V two. Now when A one is greater than A two, then the velocity V one should be less than V two. This is what is known as a nozzle that means the velocity increases in the direction of flow as the cross sectional area reduces in the direction of flow this is what is known as nozzle.

And when reverse case, when A one is less than A two that means cross sectional area increases in the direction of flow then obviously according to the conservation of mass equation. The velocity should reduce in the direction of flow and this is what is known as a diffuser.

(Refer Slide Time: 00:09:38 min)



Indian Institute of Technology, Kharagpur

Conservation of linear momentum

- Mathematical expression for Newton's second law applied to a control volume
- General equation in integral form is:

$$\left.\frac{dP}{dt}\right)_{\text{control volume}} = \frac{\partial}{\partial t}\int_{CV} v\rho\, d\forall + \int_{CS} v\rho\vec{V}\bullet d\vec{A} = \left.\vec{F}\right)_{\text{on control volume}}$$

and

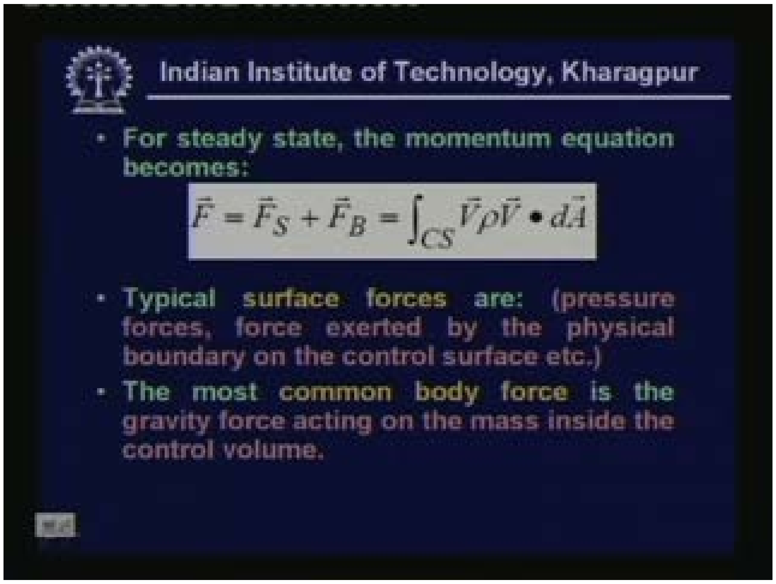
$$\left.\vec{F}\right)_{\text{on control volume}} = \sum\vec{F}_S + \sum\vec{F}_B = \frac{\partial}{\partial t}\int_{CV} v\rho\, d\forall + \int_{CS} v\rho\vec{V}\bullet d\vec{A}$$

Now let us look at the conservation of linear momentum this is nothing but a mathematical expression for Newton's second law applied to a control volume and the general equation in integral form is like this so here what we have is d P by d t applied over a control volume this term. This term is nothing but rate of change of linear momentum of the control volume and this is equal to according to Newton's second law is a net force acting on the control volume this is nothing but the Newton’s second law. And when you apply this to a control volume we can write this and introduce two terms.

You have one term here another term here. This first term accounts for the rate at which the momentum of the control volume is increasing with time. So this term accounts for the rate of change of momentum of the control volume. And the second term accounts for the net momentum flux out of the control volume. So see in simple terms the conservation of linear momentum says that the net force acting in a system is equal to sum of the rate at which the momentum of the control volume is increasing plus net momentum flux out of the control volume.

This is a vector equation. That means you can write this equation for different directions. For example you can have linear momentum equation for x direction, linear momentum equation for y direction and z direction. So this can be written in this form. The net force acting on the control volume can be split into sigma F s plus sigma F b where this is the sum total of all the surface forces and this is sum total of all body forces acting on the control volume. So finally the linear momentum equation boils down to sigma F s plus sigma F b is equal to momentum rate of momentum change into the control volume plus net momentum flux out of the control volume.

(Refer Slide Time: 00:11:33 min)



Indian Institute of Technology, Kharagpur

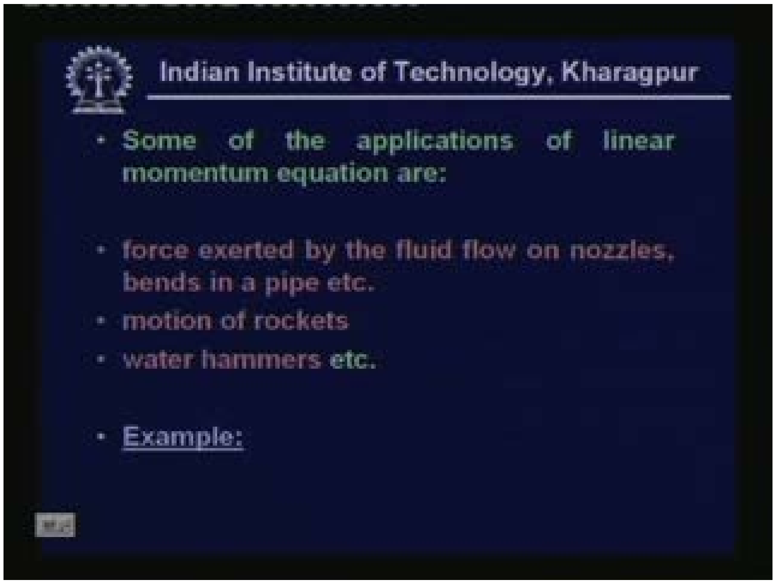
- For steady state, the momentum equation becomes:

$$\vec{F} = \vec{F}_S + \vec{F}_B = \int_{CS} \vec{V} \rho \vec{V} \bullet d\vec{A}$$

- Typical surface forces are: (pressure forces, force exerted by the physical boundary on the control surface etc.)
- The most common body force is the gravity force acting on the mass inside the control volume.

Now for steady state again just like conservation of mass for steady state you do not have the time terms. So the linear momentum equation for steady state becomes like this, where F s stands for the surface forces acting on the control surface and F b accounts for all the body forces acting on the control volume. So the sum of these two forces is equal to net momentum flux out of the control volume and what are the typical surface forces the typical surface forces are pressure forces exerted by the physical boundary on the control surface etcetera. Okay. And the most common body force is the gravity force acting on the mass inside the control volume.

(Refer Slide Time: 00:12:19 min)

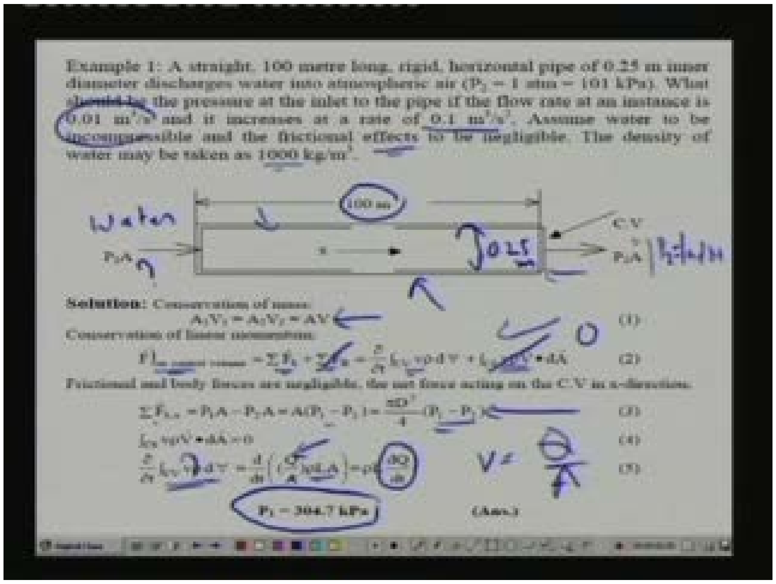


Indian Institute of Technology, Kharagpur

- Some of the applications of linear momentum equation are:
- force exerted by the fluid flow on nozzles, bends in a pipe etc.
- motion of rockets
- water hammers etc.
- Example:

Now some of the applications of linear momentum equation are force exerted by the fluid flow on nozzles bends in a pipe etcetera. That means you want to find out what is the force exerted due to fluid flow on nozzles and on the walls of the pipe and the bends etcetera then you have to apply the linear momentum equation. A linear momentum equation is also applied to find the motion of rockets in the calculation of motion of rockets and it is also applied for in water hammers etcetera.

(Refer Slide Time: 00:12:51 min)



Example 1: A straight, 100 metre long, rigid, horizontal pipe of 0.25 m inner diameter discharges water into atmospheric air ($P_2$ = 1 atm = 101 kPa). What should be the pressure at the inlet to the pipe if the flow rate at an instance is 0.01 $m^3/s$ and it increases at a rate of 0.1 $m^3/s^2$. Assume water to be incompressible and the frictional effects to be negligible. The density of water may be taken as 1000 $kg/m^3$.

**Solution:** Conservation of mass:

$$A_1V_1 = A_2V_2 = AV \quad (1)$$

Conservation of linear momentum:

$$F|_{\text{on control volume}} = \sum F_S + \sum F_B = \frac{\partial}{\partial t}\int_{CV} V\rho\, d\forall + \int_{CS} V\rho V \cdot dA \quad (2)$$

0

Frictional and body forces are negligible, the net force acting on the C.V in x-direction:

$$\sum F_{S,x} = P_1A - P_2A = A(P_1 - P_2) = \frac{\pi D^2}{4}(P_1 - P_2) \quad (3)$$

$$\int_{CS} u\rho V \cdot dA = 0 \quad (4)$$

$$\frac{\partial}{\partial t}\int_{CV} u\rho\, d\forall = \frac{d}{dt}\left(\frac{Q}{A}\rho L A\right) = \rho L \boxed{\frac{dQ}{dt}} \quad (5)$$

$V = \frac{Q}{A}$

$\boxed{P_1 = 304.7 \text{ kPa}}$ (Ans.)

Now let me give an example of the application of linear momentum equation. This is a very simple example.

What we have here, okay, let me read the problem. We have here a straight rigid hundred meter long pipe through which water flows, okay, for water flows through this pipe. And it has a diameter of point two five meters. Okay. So diameter of the pipe is point two five meter it is rigid it is horizontal and its hundred meters long. And the water enters at certain pressure and it leaves at a pressure that is equal to atmospheric pressure. That means here you have P two is equal to one atmosphere.

So basically the pipe discharges water into atmospheric pressure and at a given instant the flow rate is point one met point zero one meter cube per second that means the flow rate of water through the pipe at a given instant is point zero one meter cube per second. But it increases at a rate of point one meter cube per Second Square. That means the flow rate through the pipe is not constant but it increases with time and the rate at which the flow rate is increasing is equal to point one meter cube per second.

And we can assume the water to be incompressible and we can also neglect frictional effects and take the density of water as hundred kg meter cube. And what we have to find out is what should be the pressure at the inlet. So that water under these conditions can flow through this pipe. So first what we do is we apply the conservation of mass even though here the flow is not steady. Since we have a rigid pipe, a rigid horizontal pipe and the water is incompressible the mass of the control volume cannot change because the volume of the control volume is constant. So the amount of mass within the control volume is constant at any given instant. So it does not change its time that means water mass enters into the control volume must leave the control volume. Okay.

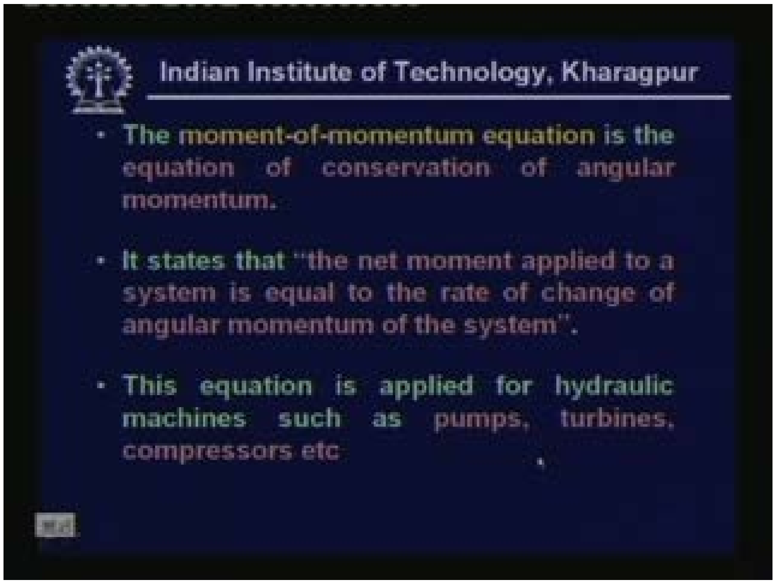
So if you apply this then you get the conservation of mass equation as A one V one is equal to A two V two. That means, and, since the area is constant at every place if its uniform cross section pipe the velocity remains constant throughout the pipe. So this is basically the conservation of mass. Now let us apply conservation of linear momentum as I have already shown conservation of linear momentum says that the net force acting on the control volume, this is the control volume, here I have shown the control volume by dash line. The net force acting on the control volume consists of the sum total of all surface forces plus sum total of all body forces. And this is equal to the rate at which the momentum of the control volume is increasing plus net momentum flux out of the control volume. That means this term is nothing but at any given instant what is the difference between the momentum out of the control volume minus momentum entering into the control volume. That means momentum at this point minus momentum at this point. Now the mass flow rate is constant at any given instant and the velocity at this point is same as velocity at this point. So their nets momentum flux out of the control volume is zero that means this term is zero for this particular problem. And since we are neglecting frictional effects and if a. since the pipe is also horizontal if there are no body force also. So this term is also not there. Then the surface forces in x direction is simply because of the pressure difference at the inlet and the outlet that means sigma F s x is nothing but P one A minus P two A that is equal to A into P one minus P two which is equal to phi D square by four into P one minus P two.

So this is the net force acting on the control volume in x direction. So this should be equal to the rate at which the momentum of the control volume is changing. So the rate at which momentum of the control volume is changing is this and here this small v is the momentum per unit area. Okay, which, I am sorry momentum per unit area. And this can also be written as its some kind of velocity and this can be written as volumetric flow rate divided by the cross section area. That means V is equal to Q by A so if you substitute Q by A for V you get this equation. And if you integrate it over the entire volume this d V is the differential volume and the total volume of the pipe is equal to L into A.

So this equation becomes d by d t Q by A into rho L A and A L A get cancelled. So ultimately you have rho L V Q by d t and in the problem it is given that d Q by d t is point one meter cube per second and rho is given as hundred kg per meter cube and length is one meter. So if we substitute these values here and then use the equations one to four you get the answer P one is equal to three hundred four point seven kilopascal absolute.

That means the pressure here should be three naught four point seven kilopascal. So that the flow can take place this is a very simple application of linear momentum to a pipe flow. And the same linear momentum principle is used in many other applications and when we discuss the fluid flow aspects through refrigerant pipes and ducts. We will discuss these issues now similar to linear momentum.

(Refer Slide Time: 00:18:17 min)

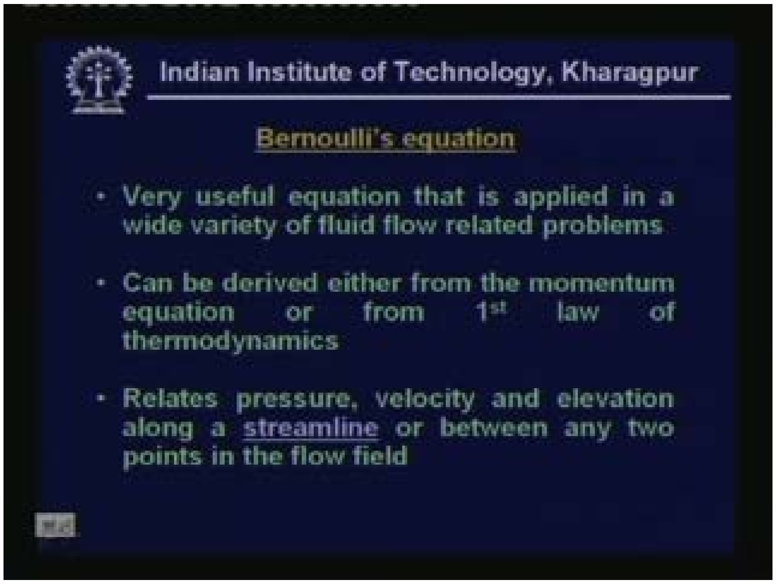


Indian Institute of Technology, Kharagpur

- The moment-of-momentum equation is the equation of conservation of angular momentum.
- It states that "the net moment applied to a system is equal to the rate of change of angular momentum of the system".
- This equation is applied for hydraulic machines such as pumps, turbines, compressors etc

We can also write a conservative equation for moment of momentum or angular momentum. This equation is known as moment of momentum equation. And this states that the net moment applied to a system is equal to the rate of change of angular momentum of the system. This is known as a conservation of angular momentum. And this is very widely used in hydraulic machines such as pumps, turbines, centrifugal compressors etcetera.

(Refer Slide Time: 00:18:42 min)



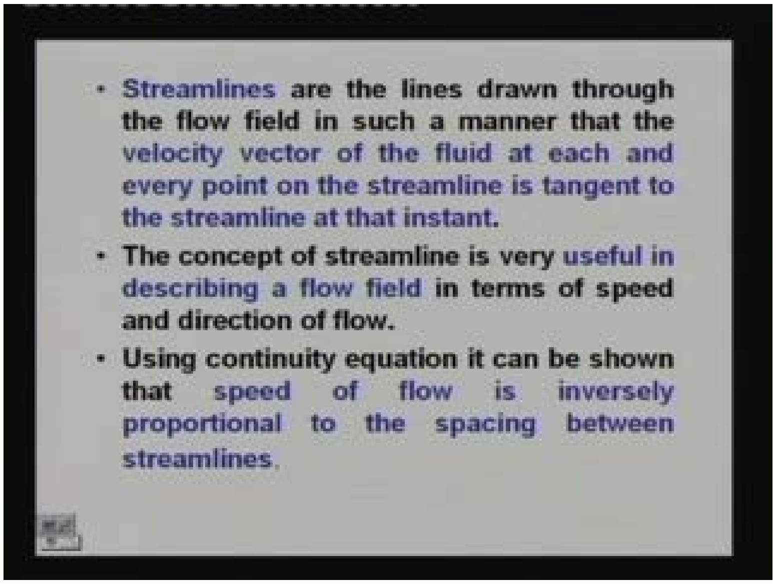
Indian Institute of Technology, Kharagpur

Bernoulli's equation

- Very useful equation that is applied in a wide variety of fluid flow related problems
- Can be derived either from the momentum equation or from 1st law of thermodynamics
- Relates pressure, velocity and elevation along a streamline or between any two points in the flow field

Now let us look at one of the very important equations called Bernoulli's equation. This equation is applied in a wide variety of fluid flow related problems and in, it can be derived either from the momentum equation or from the first law of thermodynamics. That means either you can use momentum equation or basically the Euler’s equation or you can also derive the same equation from first law of thermodynamics or conservation of energy equation. And this equation in simple terms relates pressure velocity and elevation along a streamline or pressure velocity and elevation between any two point in the flow field. Now I am sure that you have studied about streamlines in fluid mechanics.

(Refer Slide Time: 00:19:25 min)



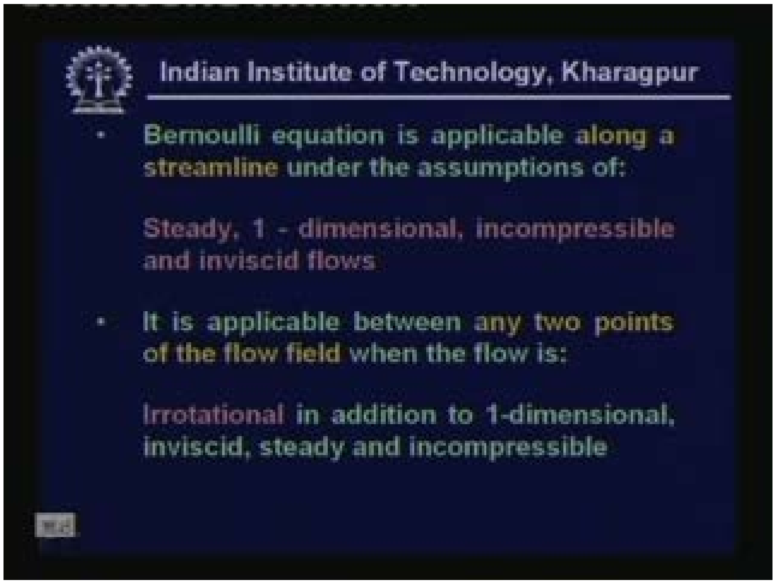
- Streamlines are the lines drawn through the flow field in such a manner that the velocity vector of the fluid at each and every point on the streamline is tangent to the streamline at that instant.
- The concept of streamline is very useful in describing a flow field in terms of speed and direction of flow.
- Using continuity equation it can be shown that speed of flow is inversely proportional to the spacing between streamlines.

Let me  just give the definition of streamlines. Streamlines are the lines drawn through the flow field in such a manner that the velocity vector of the fluid at each and every point on the streamline is tangent to the streamline at that instant. That means streamlines are lines drawn through the flow field. If you take a point on the streamline and draw tangent that gives the direction of the velocity of the fluid at that point. So this is the basic definition of streamline.

Now the concept of streamline is very useful in describing a flow field in terms of speed and direction of flow, if want to describe a flow field as you know that it is a vector field you have to specify both the magnitude and direction.

So the concept of streamline is very useful for such flow field description and it can be shown from continuity equation that when the streamlines are very close to each other. That means speed of flow is high that we or in other words speed of flow is inversely proportional to the spacing between streamlines.

(Refer Slide Time: 00:20:26 min)



Indian Institute of Technology, Kharagpur

- Bernoulli equation is applicable along a streamline under the assumptions of:

Steady, 1 - dimensional, incompressible and inviscid flows

- It is applicable between any two points of the flow field when the flow is:

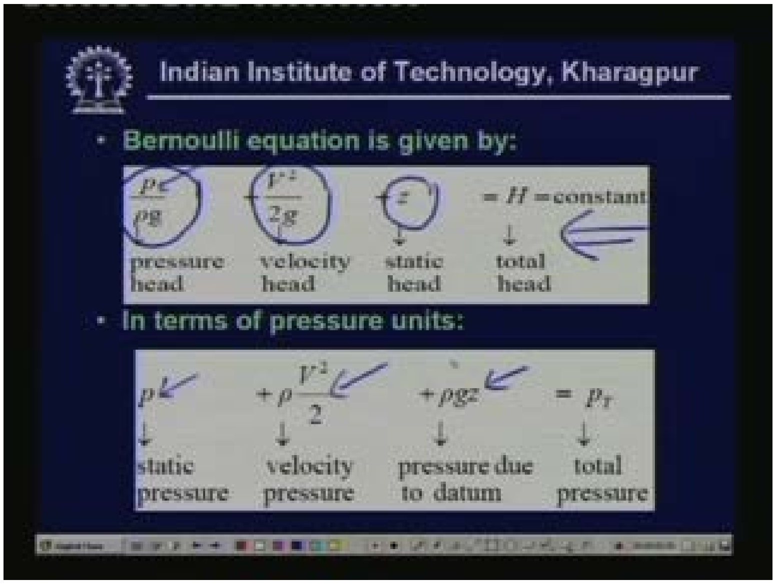
Irrotational in addition to 1-dimensional, inviscid, steady and incompressible

Now Bernoulli equation is applicable along a streamline under the assumptions of steady flow one dimensional flow incompressible flow and inviscid flow. So what do you mean by steady flow? As you know steady flow means it does not the flow, does not vary with time. So whenever you have fluid flow situation which do not vary with time you call that flow as steady flow. And what do you mean by one dimensional flow? One dimensional flow means properties vary only in one dimension. In the other two dimensions they remain constant. That means at any cross section the fluid properties remain constant.

If you are talking about one dimensional flow in y x direction that means in y and z directions there are no variations in the fluid properties at any x. So this is the meaning of one dimensional and what do you mean by inviscid? Inviscid means the fluid should not have any viscosity that means it is basically apply applicable to ideal fluid. Because all real fluids have viscosity so the moment you say that it is applicable to inviscid flow. That means it is applicable to ideal fluids and finally incompressible as we have seen means that the density does not vary along the fluid flow direction.

So under these assumptions you can apply Bernoulli equation to any two points on a streamline and the Bernoulli equation also applicable to any two points of the flow field when the flow is irrotational in addition to one dimensional inviscid steady and incompressible. What it means is if the flow is irrotational in addition to steady incompressible and inviscid. Then you can also apply Bernoulli's equation to any two points in the flow field is not necessarily on the same streamline. What do you mean by irrotational flow? Irrotational flow means the fluid particles do not undergo net rotation. So that is what is known as irrotational flow.

(Refer Slide Time: 00:22:24 min)



Indian Institute of Technology, Kharagpur

- Bernoulli equation is given by:

$$\frac{p}{\rho g} + \frac{V^2}{2g} + z = H = \text{constant}$$

| | | | |
|---|---|---|---|
| $\frac{p}{\rho g}$ | $\frac{V^2}{2g}$ | $z$ | $H$ |
| pressure head | velocity head | static head | total head |

- In terms of pressure units:

$$p + \rho\frac{V^2}{2} + \rho g z = p_T$$

| | | | |
|---|---|---|---|
| $p$ | $\rho\frac{V^2}{2}$ | $\rho g z$ | $p_T$ |
| static pressure | velocity pressure | pressure due to datum | total pressure |

Now the Bernoulli equation is given by this expression you can write it in different forms.

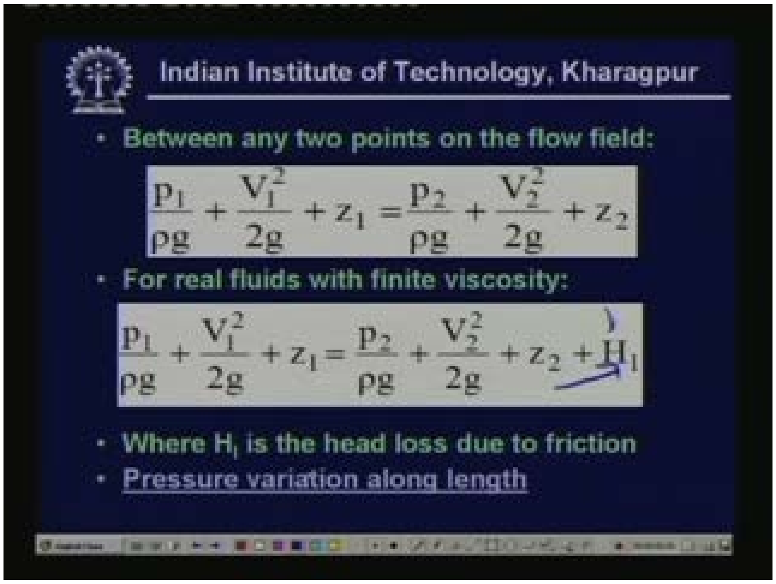
So for example if you write it in the form of head, that means in the form of pressure head velocity head static head and total head you get this kind of a an expression. Here this term as you can see is known as pressure head and this term are known as velocity head and this term is known as static head. And here P is the static pressure rho is the density and g is the acceleration due to gravity and V is the velocity of the fluid and z is the elevation with reference to a datum.

So basically what the Bernoulli equation written in terms of the head says that the pressure head plus velocity head plus static head between any two points in the flow field or between any two points in its streamline is constant. That means the total head is constant.

You can also write this equation in terms of pressures that means you can write this in terms of pressures as static pressure P velocity pressure rho V Square by two and pressure due to datum.

So Bernoulli equation when written for in terms of pressure states that, the static pressure plus velocity pressure plus pressure due to datum called as total pressure remains constant under the assumptions stated earlier.

(Refer Slide Time: 00:23:43 min)



Indian Institute of Technology, Kharagpur

- Between any two points on the flow field:

$$\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+z_2$$

- For real fluids with finite viscosity:

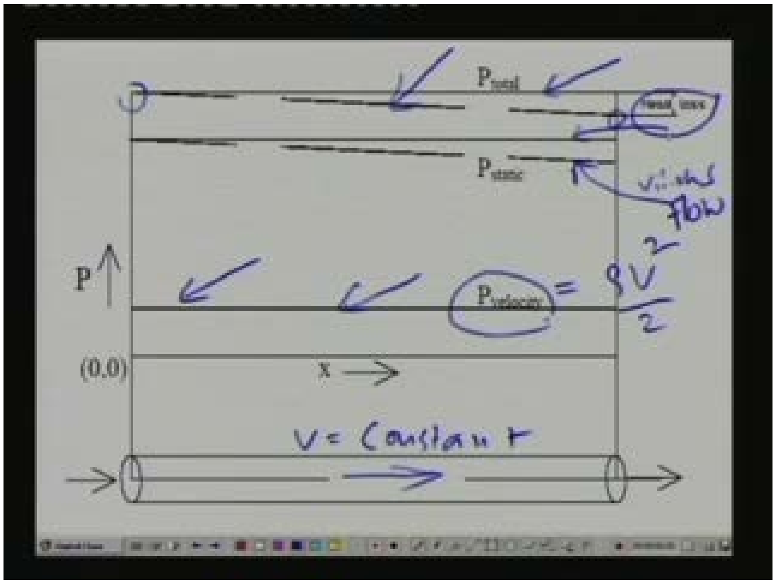
$$\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+z_2+H_l$$

- Where $H_l$ is the head loss due to friction
- <u>Pressure variation along length</u>

Now between any two points in the flow field you can write Bernoulli equation in this form, for example I am writing this between point one and two. Then the summation of static head plus velocity head plus, head due to datum is equal to the static head plus velocity head plus datum head at point two. And as I was mentioning for real fluids the flow is not inviscid. That means viscous effects are there in real fluid so you can modify Bernoulli equation and the by including what is known as a head loss term.

So this is known as modified Bernoulli equation that means it is nothing but Bernoulli equation with provision for viscosity by including head loss term. So in this expression all these terms are known to you one additional term is H l which is called as head loss. So now let me show the a pressure variation let us apply the Bernoulli equation to a pipe flow and let's see how the pressures are varying. Let us say what we have here is a uniform cross section pipe through which a fluid is flowing. Okay

(Refer Slide Time: 00:24:45 min)



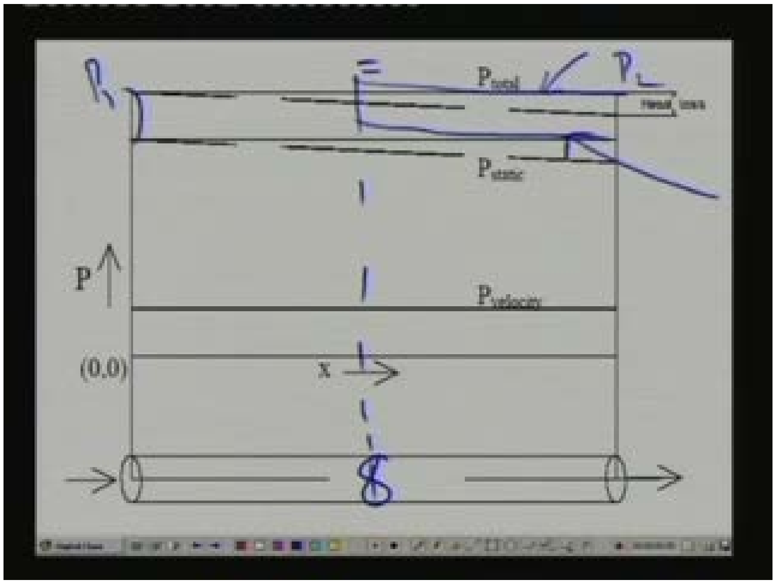
And we would like to plot different pressures. Now let us further to start with, let us assume that the flow is inviscid that means the fluid does not have any viscosity. Since the cross section area is same here. That means area at any point is same. So if we apply continuity equation at any point V is constant. So velocity does not change along the length since velocity pressure is equal to rho V square by two and rho is constant here velocity pressure is also will remain constant because V is constant.

So you if you are plotting pressure different pressures versus length you find that for the velocity pressure you get a horizontal line. Okay. So this is the line for velocity pressure. And if the fluid is ideal then the static pressure also remains same. So this is the static pressure line for the ideal fluid since the total pressure is summation of velocity pressure plus static pressure the total pressure line is also horizontal. So total pressure at any point is equal to velocity pressure plus static pressure and it remains constant throughout the length. So this is the application of Bernoulli's not to an ideal fluid now what is the effect if viscosity is non zero that means if the fluid is viscid or it has finite viscosity.

For viscous flow the velocity pressure still remains same because from the continuity equation velocity at any point should remain same. Okay. So this horizontal line applies to both inviscid as well as viscous flows. So velocity pressure remains horizontal only then what happens to the static pressure. Static pressure continuously reduces in the direction of flow because of the viscous effects that means there is a loss of static head  due to viscosity so you see the  a sloped dash line for viscous flow. Okay.

So the total pressure is nothing but the summation of static pressure plus velocity pressure so and the velocity pressure is remaining constant and static pressure is reducing. So the total pressure also reduces in a real flow and this difference between the total pressures at the inlet minus total pressure at the outlet is nothing but your head loss. So this is the application of Bernoulli equation to a pipe flow.

(Refer Slide Time: 00:27:21 min)

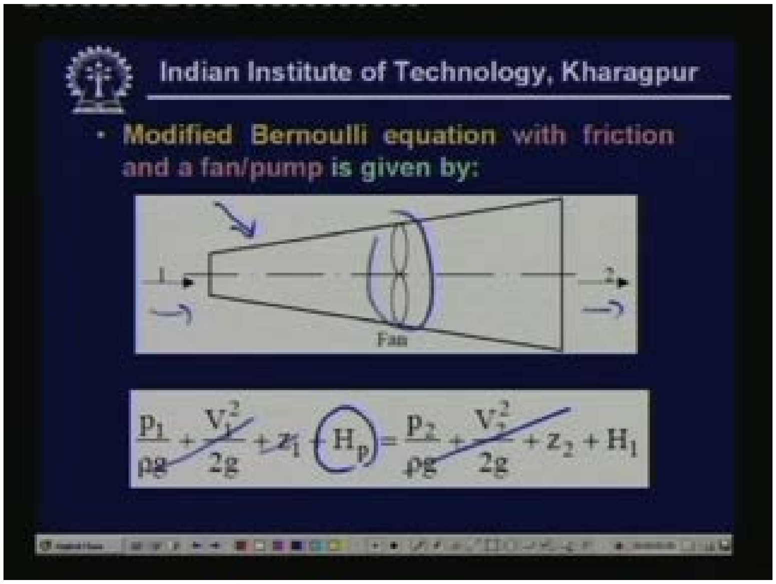


Now let us say that we would like to maintain the same pressure at the inlet and as well as at the outlet and the flow is inviscid. That means we want to maintain P one should be same as P two and similarly static pressure the inlet should be same as static pressure at the outlet. But the fluid is viscid. That means it has viscosity. Then how do we ensure this? So whenever you have viscous flows and if you want to have same pressures at the inlet or outlet or if the inlet pressures are greater than the outlet pressures then what we have to do is we have to use some kind of a pump or a fan for example if i am using the pump here or a fan here. Okay.

Then it is possible to maintain the same static pressures and the total pressures at the inlet and the outlet. What happens when you have, when you are putting a pump, when you are keeping a pump or a fan in the pipe flow the velocity pressure does not change because it  the fan or pump does not increase the mass flow rate so velocity pressure remains same. But it increases the static pressure because it adds energy to the fluid so at the point where you have added the pump the static pressure increases.

Okay. Since the starting pressure increases the velocity pressure also increases. So you have the modified pressure lines with the addition of pump or fan inside the fluid circuit.

(Refer Slide Time: 00:28:58 min)



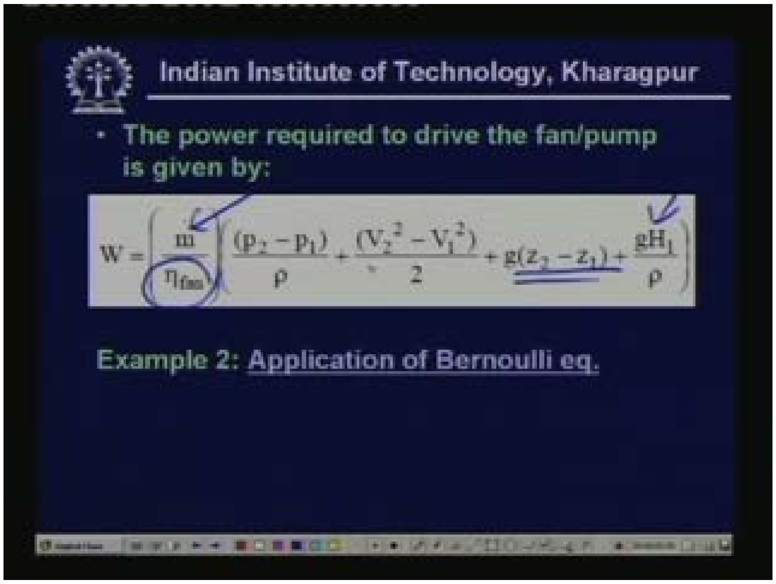
Indian Institute of Technology, Kharagpur

- Modified Bernoulli equation with friction and a fan/pump is given by:

$$\cancel{\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+z_1}+H_p=\cancel{\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+z_2}+H_l$$

Now that is what is shown in this line. This is the modified Bernoulli equation with friction and a fan or pump. And I have shown here an arbitrary cross section that means cross section area and inlet to constant and the fluid enters here and fluid leaks here. And this is the fan or a pump so what is the modified Bernoulli equation for this kind of situation this is the modified Bernoulli equation only the difference is this. So this is nothing but the head gain due to the presence of the fan if p one is equal to one by rho g is equal to p two by rho g V one square by two g is equal to V two square by two g and z one is equal to z two then these terms get cancelled out. And you have H p is equal to H loss that means the head gain due to the fan should be sufficient to overcome the head loss due to friction.

(Refer Slide Time: 00:29:56 min)



Indian Institute of Technology, Kharagpur

- The power required to drive the fan/pump is given by:

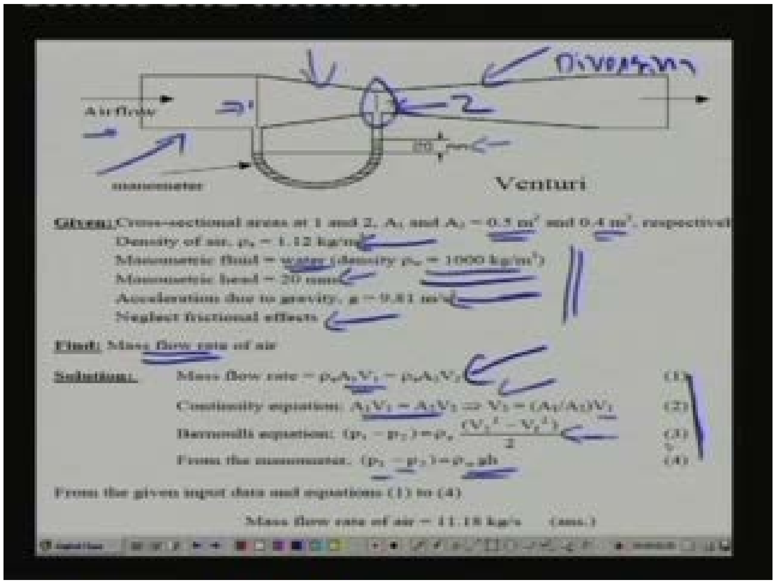
$$W = \left(\frac{\dot{m}}{\eta_{fan}}\right)\left(\frac{(p_2 - p_1)}{\rho} + \frac{(V_2^2 - V_1^2)}{2} + \underline{g(z_2 - z_1)} + \frac{gH_1}{\rho}\right)$$

Example 2: Application of Bernoulli eq.

Now what is the power required to drive the fan or pump under the case discussed just now so this equation gives what is the power requirement. When you are using a fan or a pump and this is again is derived by using Bernoulli's equation and what we have here this m dot is nothing but the mass flow rate of the fluid. And this is the head loss and this is the static pressure at the outlet the static pressure at the inlet V two square minus V one square is the difference divided by two is the difference in the velocity pressures at the outlet and inlet.

And this is the difference in the datum heads and here eta fan is the efficiency of the fan that means if the fan has some inefficiency then we have to take that into account. So this expression gives you the fan power required or pumps power required. So you can by applying Bernoulli’s equation you can find out what should be the motor capacity to drive the fan or pump. Now let us apply this Bernoulli equation to a very simple case.

(Refer Slide Time: 00:31:00 min)



**Given:** Cross-sectional areas at 1 and 2, $A_1$ and $A_2$ = 0.5 m$^2$ and 0.4 m$^2$, respectivel

Density of air, $\rho_a$ = 1.12 kg/m$^3$

Manometric fluid = water (density $\rho_w$ = 1000 kg/m$^3$)

Manometric head = 20 mm

Acceleration due to gravity, g = 9.81 m/s$^2$

Neglect frictional effects

**Find:** Mass flow rate of air

**Solution:** Mass flow rate = $\rho_a A_1 V_1 = \rho_a A_2 V_2$ (1)

Continuity equation: $A_1 V_1 = A_2 V_2 \Rightarrow V_2 = (A_1/A_2) V_1$ (2)

Bernoulli equation: $(p_1 - p_2) = \rho_a \dfrac{(V_2^2 - V_1^2)}{2}$ (3)

From the manometer, $(p_1 - p_2) = \rho_w gh$ (4)

From the given input data and equations (1) to (4)

Mass flow rate of air = 11.18 kg/s (ans.)

So this is an application of Bernoulli equation to a Venturi now. What is the Venturi? Venturi is a device for measuring flow rates basically normally it is used for measuring air flow rate or water flow rate. So as you can see in the figure it consists of a straight portion then a converging portion and a gradually diverging portion. So this is a diverging portion this is the converging and where the diverging and converging portions meet we call as that portion as the throat of the Venturi.

And here this is the inlet to the converging portion is point one and the throat portion is point two and Venturi has a manometer here. And one end of the manometer is connected to the inlet to the converging section. That means point one and the other end of the manometer is converging is connected to the throat portion that means point two. And this manometer is filled with some manometric liquid. So basically the air flow through the manometer and because the air flow there will be some pressure difference in the manometer. That means there will be a manometric head here.

And from the characteristics of the Venturi and from the by measuring the manometric head you can calculate the air flow by applying Bernoulli equation. So now let us look at the problem statement it is given here that the area of cross section area A one and A two are point five meter square and point four meter square. That means area at this point one is point five meter square and area at point two is point four meter square. And the density of air is one point one two kg per meter cube and manometric fluid used here is water and it has a density of thousand kg per meter cube.

And the measurement shows that the manometric head is twenty mm and it is also given that acceleration due to gravity is nine point eight one meter per Second Square. And we can neglect all frictional effects. That means we will be applying the original Bernoulli equation because we are assuming the fluid to be incompressible steady irrotational and inviscid. So frictional effect need not be considered so we apply a Bernoulli equation let us see what happens. First what is the, okay, what is the required output we have to find out put what the mass flow rate is for this Venturi which shows a manometric head of twenty mm?

So the equation for mass flow rate is rho A V mass flow rate at any point is equal to rho into cross sectional area at that point and velocity at that point. So you can write this expression for mass flow rate. And from continuity equation nothing but conservation of mass equation A one V one is equal to A two V two because we are assuming the fluid to be incompressible. So rho A at any point is constant so from this equation you get that A one V one is equal to A two V two or velocity at the point two. That means velocity at the throat can be expressed in terms of the areas cross sectional areas at the throat and at the inlet to the converging section and in and velocity at the inlet to the converging section.

So V two is equal to A one by A two into V one. Now let us apply Bernoulli equation if you are applying the Bernoulli equation you can slightly write it in the different form datum head is change in datum head is zero. So we can write the Bernoulli equation in this form P one minus P two is equal to rho A into V two square minus V one square by two and P one minus P two is nothing but the pressure difference between the points one and two which can be obtained from the manometer reading as rho g h. Where rho, W here is the density of the water g is acceleration due to gravity and h is the manometric head.

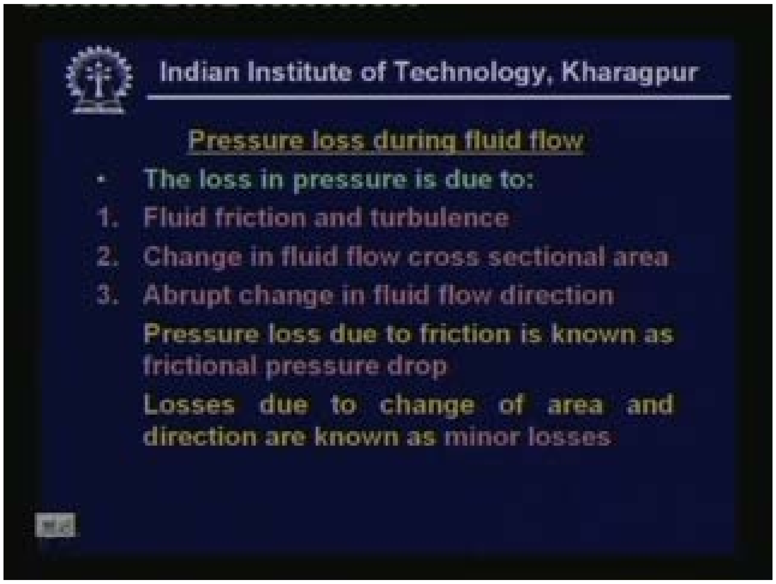
So if you are use the input data and these four equations you can you will find that the mass flow rate of air is equal to eleven point one eight kg per second so this is one very useful application of Bernoulli equation to a Venturi meter.

There are many other applications of Bernoulli equation as I mentioned it is one of the most useful equations and you will find its applications particularly in air conditioning duct design and in refrigeration piping design etcetera will be using this equation repeatedly. But you must keep certain things in mind even though Bernoulli equation is very useful you must be aware of the assumptions under which the Bernoulli equation is valid.

For example Bernoulli equation is not valid when you have a duct where there is a sudden expansion when there is when there is a sudden expansion what happens is the fluid gets separated that means there will be separation of fluid. And the flow no longer remains one dimensional so you cannot apply Bernoulli equation because you remember that Bernoulli equation is a scalar equation and it's applicable to one dimensional flow. So whenever you have two dimensional three dimensional effects you cannot apply Bernoulli equation number one and second thing is if you have a fluid flow where there is lot of heat transfer as the fluid flows through the duct then also you cannot apply Bernoulli equation. Then because what happens when there is the large amount of heat transfer then the density may not remain constant as the fluid is flowing through the conduit.

That means the assumption of incompressible flow will not be valid so you cannot again apply the simple Bernoulli equation that we have discussed here. So when you are applying Bernoulli equation first make sure that the assumptions or the conditions are met and then apply the Bernoulli equation. Of course Bernoulli equation are also have been developed for unsteady flows and also for compressible flows but we will be not discussing those issues these will be discussed in applied fluid mechanic courses.

(Refer Slide Time: 00:37:00 min)



Indian Institute of Technology, Kharagpur

Pressure loss during fluid flow

- The loss in pressure is due to:

1. Fluid friction and turbulence
2. Change in fluid flow cross sectional area
3. Abrupt change in fluid flow direction

Pressure loss due to friction is known as frictional pressure drop

Losses due to change of area and direction are known as minor losses

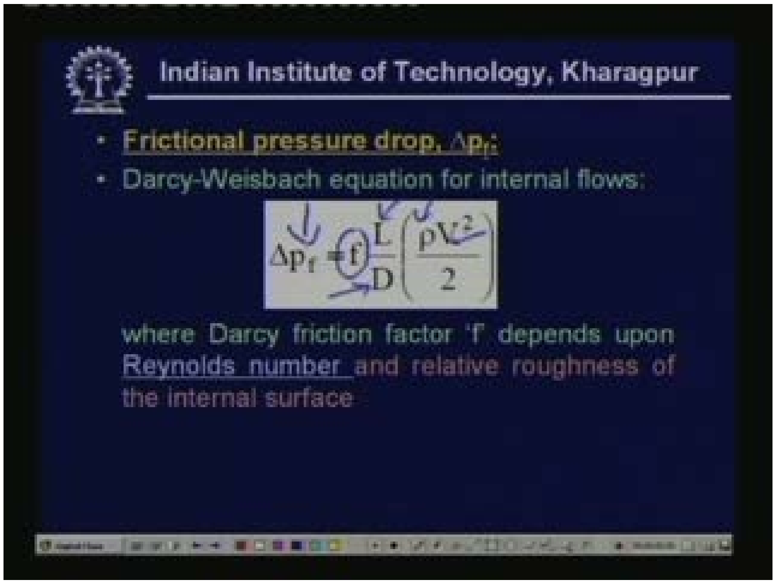
Now let us look at the evaluation of pressure loss during fluid flow. We have seen that generally in one of the most common problems or common requirements in the design of an air conditioning that or in the design of a refrigerant piping or in a chill water piping is to find the capacity of the motor or capacity of the pump. What should be the power consumption or what is the power consumption of the pump. So that you can select a suitable motor for the pump or the fan. So you have seen from the modified Bernoulli equation in the presence of viscous effects and the fan that the pump power depends upon the mass flow rate and it depends upon the inlet and outlet pressures velocities and datum heads and the friction losses that means the head loss.

So if you want to find out  what is the power requirement you must know what is the head loss okay, and normally in any in any given problem the mass flow rate is an input and the pressures and the velocities also in  are also inputs.

So if you know the head loss then you can find out the power input. So the key problem here is to find out what is the pressure drop as the fluid flows through a conduit. Now the loss in pressure is due to fluid friction and turbulence change in fluid flow cross sectional area and abrupt change in fluid flow direction. That means the fluid undergoes pressure loss due to these three effects friction and turbulence change in fluid flow cross sectional area and abrupt change in fluid flow direction. A pressure loss due to friction and turbulence is known as frictional pressure drop.

And pressure drop due to change in cross sectional area and abrupt change in fluid flow direction is known as minor loss. That means frictional losses all losses due to friction and turbulence are known as frictional pressure drops and losses due to change in cross sectional area and change in direction are known as minor losses. Sometimes name minor losses could be a misnomer because in some situations the minor losses can be must more than the frictional pressure drops. So you have to keep it in mind and you should not neglect that the minor loss is thinking that they are minor.

(Refer Slide Time: 00:39:25 min)



Indian Institute of Technology, Kharagpur

- Frictional pressure drop, $\Delta p_f$:
- Darcy-Weisbach equation for internal flows:

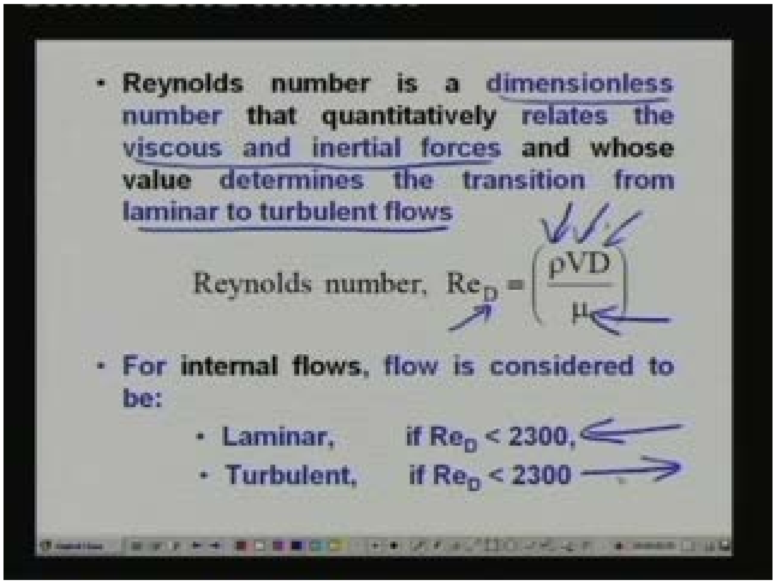
$$\Delta p_f = f \frac{L}{D}\left(\frac{\rho V^2}{2}\right)$$

where Darcy friction factor 'f' depends upon Reynolds number and relative roughness of the internal surface

Now the frictional pressure drop the frictional pressure drop is given by Darcy Weisbach equation for internal flows and it is given as delta p f is equal to f L by D into rho V square by two this is the frictional pressure drop. And here L is the length of the pipe or tube or the duct and D is the internal diameter if it is circular pipe or tube or by an equivalent hydraulic diameter. If the cross section is non circular and rho as you know is the density and V is the velocity. So if you know the density velocity length diameter and this factor then you can find out the frictional pressure drops.

Now what is the factor f? Factor f is known as Darcy friction factor it’s a non dimensional factor and it depends upon Reynolds number and relative roughness of the internal surface. Now what is Reynolds number? I am sure that again you studied about Reynolds number in your course and fluids mechanics.

(Refer Slide Time: 00:40:30 min)



- Reynolds number is a dimensionless number that quantitatively relates the viscous and inertial forces and whose value determines the transition from laminar to turbulent flows

Reynolds number, $Re_D = \left(\frac{\rho VD}{\mu}\right)$

- For internal flows, flow is considered to be:
  - Laminar, if $Re_D < 2300$,
  - Turbulent, if $Re_D < 2300$

But let me just give it for the sake of completion Reynolds number is a dimensionless number that quantitatively relates the viscous and inertial forces and whose value determines the transition from laminar to turbulent flow. So basically you have to keep it in mind that this is the dimensionless number and it relates viscous and inertial forces and its value determines whether the flow is laminar or turbulent we will see litter later what is a mean by laminar or turbulent flow. And mathematically Reynolds number is expressed as R e is equal to rho V D by mu where rho is the density V is the fluid velocity and D is the length scale parameter if it is a tube or circular tube or pipe D is the diameter.

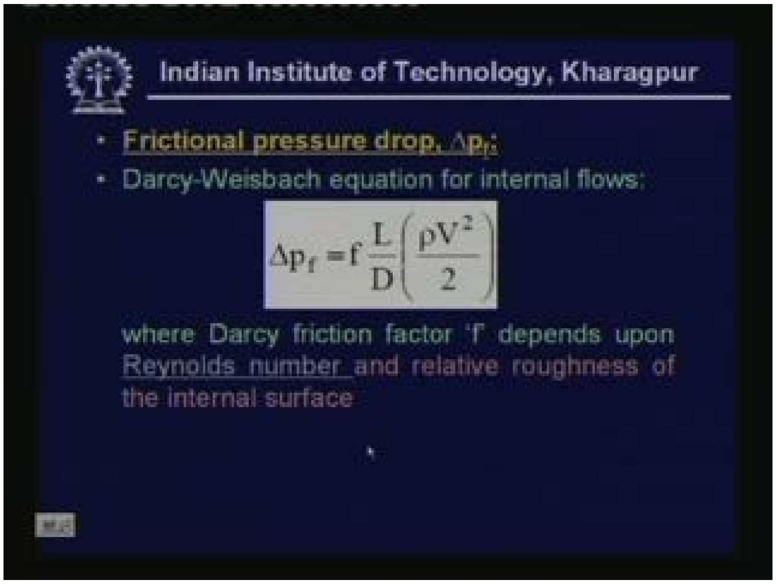
If it is a non circular tube or pipe D can be could be a hydraulic diameter and if it is a external flow that means the flow is taking place over a flat plate something like that then D can be the length and mu here is the dynamic viscosity. So Reynolds number is nothing but rho V D by mu and for internal flows it is observed that the flow will be laminar if Reynolds number is less than two thousand three hundred and the flow becomes turbulent if the Reynolds number increases beyond two thousand three hundred.

Now this is only a very rough criteria strictly speaking you really do not have a sudden transition from laminar flow to turbulent flow. That means basically we will see, I will show you the boundary layer development generally you have at low Reynolds number you have laminar flow and at a certain critical Reynolds number there will be transition. So that means flow turn changes from laminar to a transition flow and the transition flow continues to a certain extent and beyond certain Reynolds number the flow becomes turbulent. That means you have laminar flow transition regime and turbulent regime. But where most of the practical purposes and most of the engineering calculations we assume that when the Reynolds number particularly for internal flows if it is less than two thousand three hundred we take that the flow is laminar and if it exceeds two thousand three hundred we assume that the flow is turbulent. But in advance the heat transfer and fluid mechanics problem the transition region is also concerned but as far as our refrigeration and air conditioning calculations are concerned you may take the transition Reynolds number or critical Reynolds number as two thousand three hundred for internal flows.

One more thing is that it is possible by carefully controlling the fluid flow conditions and all to extend the critical Reynolds number. That means you can have laminar flow even at Reynolds number as high as say ten thousand or twenty thousand under carefully controlled conditions.

So this critical Reynolds number and the number two thousand three hundred is a rough guide line it is not a fixed number or any thing.

(Refer Slide Time: 00:43:32 min)



Indian Institute of Technology, Kharagpur

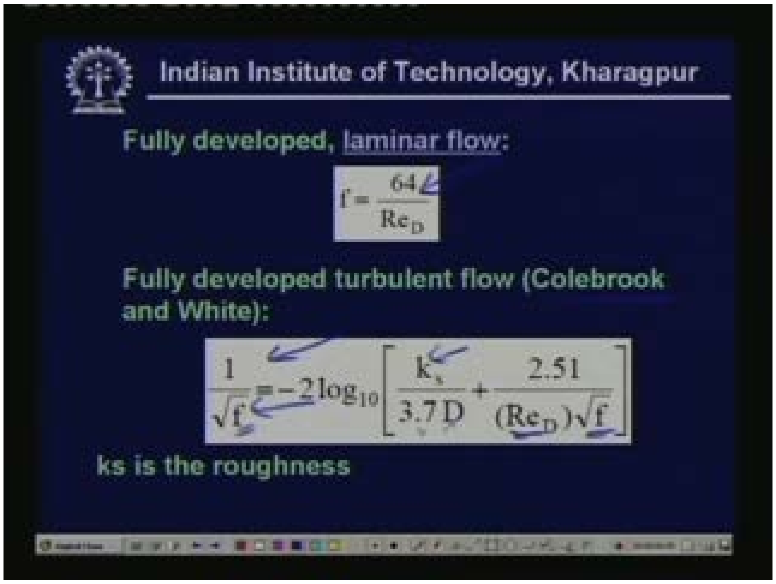
- Frictional pressure drop, $\Delta p_f$:
- Darcy-Weisbach equation for internal flows:

$$\Delta p_f = f\frac{L}{D}\left(\frac{\rho V^2}{2}\right)$$

where Darcy friction factor 'f' depends upon Reynolds number and relative roughness of the internal surface

So we have, I have mentioned that the Darcy friction factor is a function of Reynolds number and relative roughness of the internal surface.

(Refer Slide Time: 00:43:43 min)



Indian Institute of Technology, Kharagpur

Fully developed, laminar flow:

$$f = \frac{64}{Re_D}$$

Fully developed turbulent flow (Colebrook and White):

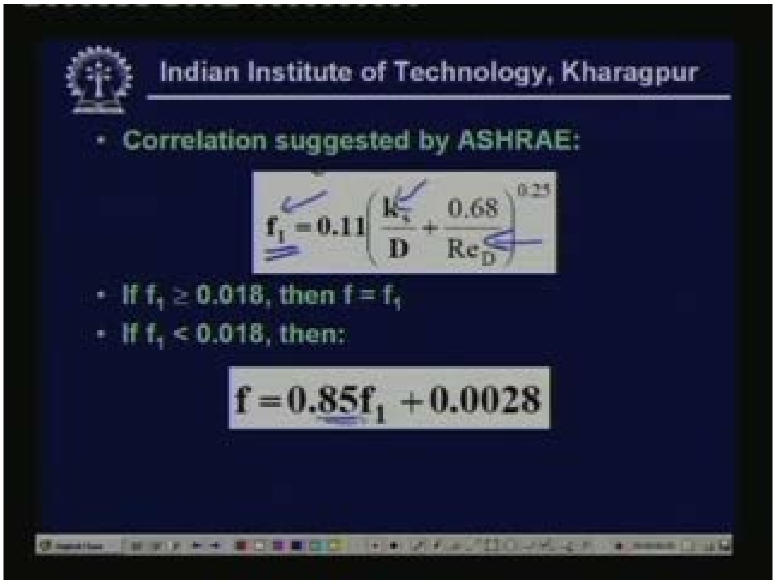
$$\frac{1}{\sqrt{f}} = -2\log_{10}\left[\frac{k_s}{3.7D} + \frac{2.51}{(Re_D)\sqrt{f}}\right]$$

ks is the roughness

So let us look at few cases for fully developed laminar flow. The Darcy friction factor is given by sixty-four divided by the Reynolds number. That means for fully developed laminar flow the friction factor is independent of the roughness of the surface if it is only a function of the Reynolds number. And for fully developed turbulent flow the correlation given here is known as Colebrook and White correlation is Colebrook and White correlation and it relates the friction factor with the roughness and the Reynolds number.

This roughness has same dimensions as the length scale or that means the diameter. So this equation is very popular equation and it is known as Colebrook and White equation for the fully developed turbulent flow. One thing you can notice here is that to solve, to use this equation you have to use an iterative procedure because f occurs both on the left hand side as well as on the right hand side. So you have to use the trial and aware iterative method to find out f.

(Refer Slide Time: 00:44:50 min)



Indian Institute of Technology, Kharagpur

- Correlation suggested by ASHRAE:

$$f_1 = 0.11\left(\frac{k_s}{D} + \frac{0.68}{Re_D}\right)^{0.25}$$

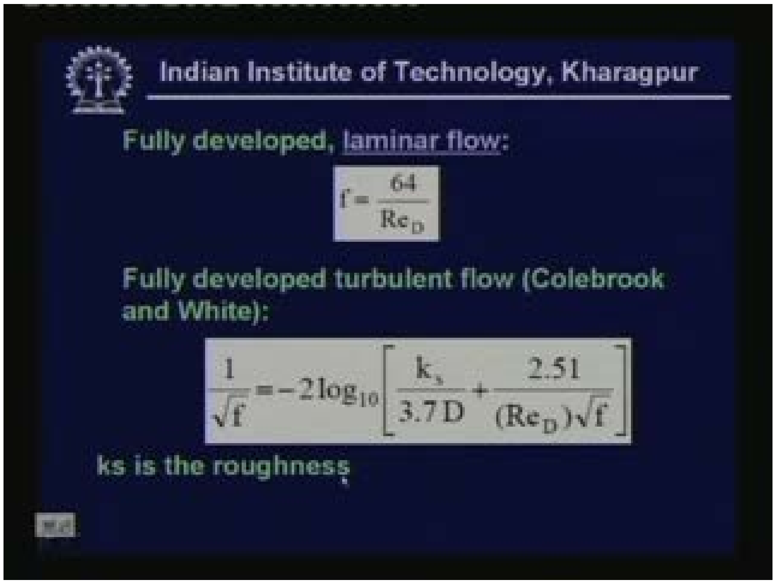
- If $f_1 \geq 0.018$, then $f = f_1$
- If $f_1 < 0.018$, then:

$$f = 0.85 f_1 + 0.0028$$

Now ASHRAE suggests another correlation very useful correlation and explicit correlation first ASHRAE suggests ASHRAE defines a parameter f one f one is given as point one into k s by D plus point six eight divided by Reynolds number. So this factor is defined first and if this factor is less than point zero one eight then f one is equal to friction factor f. And if the factor is less than point zero one eight then friction factor is given by this equation. That means first you have to find out the factor f one from the roughness and from the Reynolds number and see whether the factor is greater than or equal to point zero one eight or it is less than point zero one eight.

If it is greater than or equal to point zero one eight then f one is nothing but f, if it is less than point zero one eight you have to make a correction and f is equal to point eight five f one plus point zero zero two eight so this is the very useful equation suggested by ASHRAE.

(Refer Slide Time: 00:46:00 min)



Indian Institute of Technology, Kharagpur

Fully developed, laminar flow:

$$f = \frac{64}{Re_D}$$

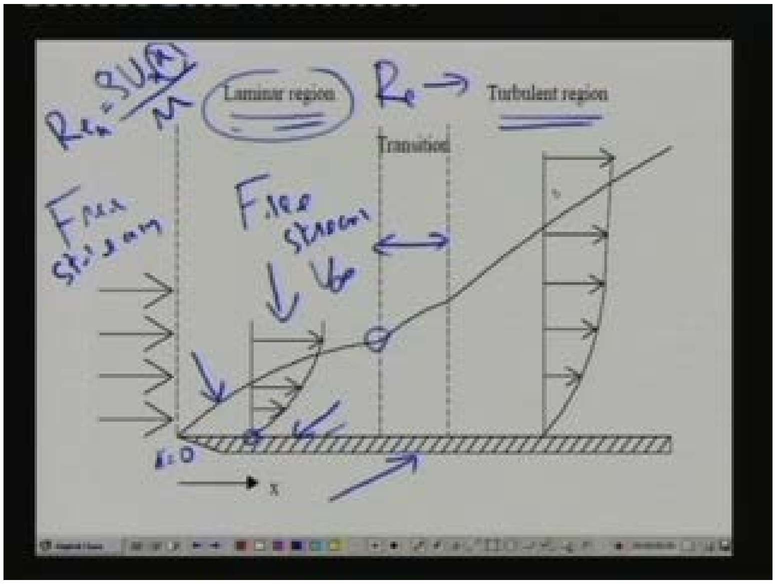
Fully developed turbulent flow (Colebrook and White):

$$\frac{1}{\sqrt{f}} = -2\log_{10}\left[\frac{k_s}{3.7\,D} + \frac{2.51}{(Re_D)\sqrt{f}}\right]$$

ks is the roughness

Now I was talking about laminar flow and turbulent flow.

(Refer Slide Time: 00:46:07 min)



Let me just explain laminar and turbulent flow and what you mean by fully developed flow?

Let us say that we have a flat plate, a flat plate over which fluid is flowing. So at this point you have what you known as a free stream and this fluid comes in contact with a solid surface at this point let us say at this point x is zero. So at this point the fluid comes in contact with the solid once the fluid comes in contact with solid what happens.

Let us consider real fluid. That means let us say that the fluids had finite viscosity as you know viscosity is one of the transport properties of the fluid and when a real fluid comes in contact with a let us say a stationary surface. Then the velocity of the fluid adjacent to the stationery surface will be same as that of the stationery surface. That means the velocity next to the solid surface will be same as the solid surface temperature solid surface velocity. And if the solid surface is stationery solid surface velocity is zero. That means the fluid layer adjacent to the solid surface will be having a zero velocity that means velocity at this point will be zero okay. So suddenly the velocity changes from the free stream velocity to zero velocity on the surface.

Once the velocity is zero you have to satisfy the continuity equation. That means the velocity profile will develop the velocity profile will develop because of viscous effects. Since the velocity here is zero the fluid layer on the surface will try to hold the fluid layer adjacent to it.

That means it will try to decelerate the fluid that is adjacent to the layer and the, that layer will try to decelerate the fluid that is flowing adjacent to it so like that a velocity gradient develops.

And that is what is shown here so gradually velocity gradient develops and at a certain point the velocity will be same as the free stream velocity and again let us say this is the free stream velocity.

This is free stream on the plate and this is the free stream velocity on the plate let us say that U infinity on the plate okay. So what basically has happened is because of the presence of the solid surface a velocity gradient has developed in this region. And the region in which the velocity varies from zero to the free stream velocity is known as the boundary layer that means this is known as the boundary layer. So because of the presence of a surface and because of viscous effects a boundary layer develops whenever a fluid comes in contact with a solid surface okay.

Now initially this boundary layer will be laminar. So what do you mean by laminar? Laminar flow means the fluid particles will be flowing in layers or in lamina and any mixing is only due to molecular motion. That means in laminar flow mixing is mixing between fluids layers are purely because of molecular motion and there is no bulk mixing such a flow is called as laminar flow.

And for laminar flow appear very smooth and it is steady and a typical example of laminar flow is the flow of let us say thick liquid like honey from a bottle that means you take a honey bottle and pour it then the honey comes out in a laminar manner okay. And laminar flow is generally encountered when the Reynolds number is small so at low Reynolds number you generally have laminar flow. So in this particular case to start with we have laminar region and for this particular case we define Reynolds number in terms of the length scale that means the local Reynolds number is defined as divided by U okay. So here this length scale is x or the distance from the leading edge so as the distance from the leading edge increases Reynolds number increases. That means Reynolds number increases in this direction. So to start with we have a laminar region where the Reynolds number is small and as I said that certain critical Reynolds number the flow changes from laminar to transition.

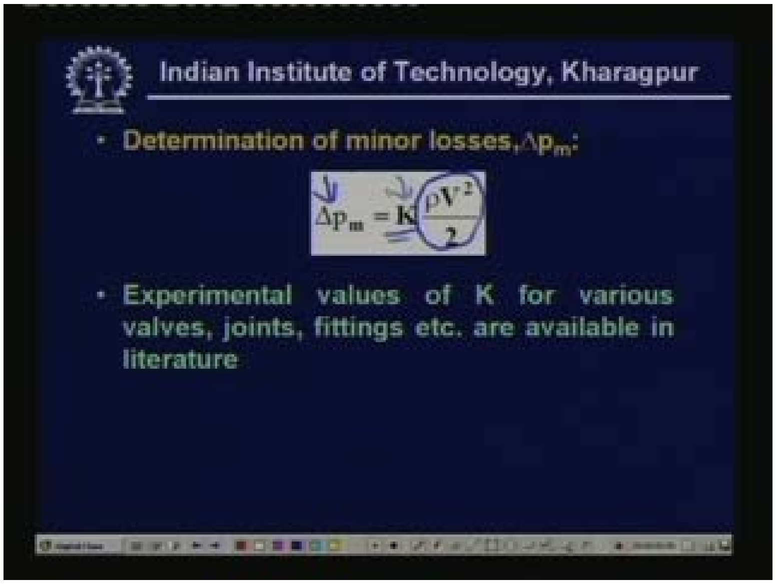
So you have a transition regime here. So transition regime continues to certain extent and again at certain point flow transition takes place to turbulent region. So you have turbulent flow. So you to start with you have laminar region you have transition region and turbulent region. What happens in a turbulent region? In a turbulent region you have molecular motion and super imposed on that molecular motion is a bulk mixing of fluid particles because of eddies. That means in turbulent flow eddies form and these eddies lead to mixing on a bulk scale. That means on a microscopic scale.

So you have mixing due to molecular motion as well as mixing due to eddies so this is the characteristics of a turbulent flow. And the unlike laminar flow turbulent flow is highly unsteady and a typical example of turbulent flow is let us say that you have a water tap and you have opened the tap fully then water comes out in very random irregular manner and that is an example of turbulent flow. And you slowly close the water tap then water becomes more and more regular and at very low flow rates the water becomes flow becomes very smooth that means you have a laminar region.

So this is how you can change the laminar region to turbulent region by changing the velocity in this case or in Reynolds number. So this is the in short the physics behind laminar and turbulent flows and as I said the value of Reynolds number decides whether the flow is laminar or turbulent and for internal flows I mentioned that if the Reynolds number is less than two thousand three hundred you have laminar flow. And if it is greater than two thousand three hundred you may consider that as turbulent flow this is only the guide line and for fluid flow over flat plate that means the example that i have shown just now. Or the critical Reynolds number is totally different so in for flow over horizontal plate the flow will be laminar as long as the Reynolds number is less than five into ten to the power of five this is again a guide line.

So as long as Reynolds number is less than five into ten to the power of five we assume that the flow is to be laminar and if exceeds if it exceeds five into ten to the power of five we assume the flow to be turbulent.

(Refer Slide Time: 00:53:10 min)



Indian Institute of Technology, Kharagpur

- Determination of minor losses, $\Delta p_m$:

$$\Delta p_m = K\frac{\rho V^2}{2}$$

- Experimental values of K for various valves, joints, fittings etc. are available in literature

So now let us look at the determination of minor losses as I mentioned minor losses are due to change in cross section and also due to change in the direction normally when the cross section increases gradually or reduces gradually there will not be any significant losses. But when the cross section increases steeply then there will be boundary layer separation and there will be losses. So these losses are known as minor losses in addition to that whenever there is a change in flow direction basically an abrupt change again if the change takes place gradually then the losses are negligible.

But if the change is abrupt for example you have the sudden elbow then there is a sudden change in the direction then you will have again pressure losses. So the losses due to change in direction and change in cross section area are clubbed together and they are known as minor losses. So generally the minor losses are given as delta p m this is a sum total of the minor losses and that is equal to K a factor K into rho V square by two that means the minor losses are always proportional to the velocity pressure term okay. And this factor K has to be determined experimentally and little bit different for different types of valves joints fittings bends etcetera.

And generally the values of K for different fittings valves etcetera are available in standard literature so what we have do if we want to find out what is the minor loss.

For example you have a bend then you would like to find out what is the pressure loss due to bend. Then first you have to find out what is the velocity pressure that means you have to find out the velocity and you have to find out the term rho m square by two and multiply that into the factor K. So this factor K has to be obtained from a published data for that particular bend or valve or fitting so this is generally the procedure for calculating minor losses. So finally the total head loss is nothing but the frictional loss plus minor loss. So we have to find out these two from that you have to find out the power required for a fan or pump which is the typical problem in any fluid flow related problems in refrigeration and air conditioning.

Now let us conclude today's lecture.

(Refer Slide Time: 00:55:36 min)

Indian Institute of Technology, Kharagpur

Conclusions

- Fundamentals of fluid flow relevant to refrigeration & air conditioning are discussed
- Mathematical equations are presented for conservation of mass and momentum
- Bernoulli equation in its original and in modified forms is discussed
- Methods for evaluating friction losses and minor losses are presented with typical correlations for friction coefficient
- The applications of the above in refrigeration and air conditioning will be discussed later

In today's lecture we have reviewed the fundamentals of fluids flow relevant to refrigeration and air conditioning and we have presented mathematical equations for conservation of mass and momentum. Conservation of mass is also known as equation for conservation of mass is also known as continuity equation both is same. And we have also presented and discussed Bernoulli equation in its original form and also modified Bernoulli equation for viscous flows and when fan or pump is used in the fluid flow conduit. Then we have presented methods for evaluating friction losses and minor losses with typical correlations for friction coefficients.

Let me mention here that the correlation shown here for friction factor are typical correlations. That means there are a large number of correlations but different fluid flow situations and these correlations are applicable generally they are empirical. So they are applicable for only that particular condition that means every correlation has a range of application. So when you are applying empirical correlations you must see what its range of applicability is and make sure that your problem falls into that range of applicability only then apply the correlation. And there are large numbers of correlations available for friction factors for different types of flows. So depending up on the particular situation we have to select a suitable correlation then find the friction factor. Similarly the minor losses we have to find out the factor K depending up on the specific problem and from these two we had finds out the total head loss. So these aspects have been discussed here and we will be applying these to refrigeration and air conditioning in a later chapter okay.

Thank you.