Fluid Dynamics

The primary objectives of this section is to introduce the student to

Basic fluid flow

July 11, 2023

Conservation laws of transport phenomena and

To learn in details the basic principles and application of fluid mechanics (momentum transport) to engineering problems.

Fluid dynamics is a branch of physics that deals with the study of fluids, which include both liquids and gases, and the motion of these substances. It involves the understanding and analysis of how fluids behave under various conditions, such as flow patterns, pressure distributions, and forces acting upon them.

When dealing with fluid dynamics, there are key principles and concepts one should always consider-and they include:

Conservation Laws: Fluid dynamics is based on the conservation laws, namely the conservation of mass, momentum, and energy. These laws state that mass, momentum, and energy cannot be created or destroyed but can only be transferred or transformed.

Continuum Hypothesis: Fluid dynamics treats fluids as continuous substances, assuming that they are composed of an infinite number of infinitesimally small particles. This approximation is valid as long as the fluid is not extremely rarefied or at the molecular scale.

Fluid Properties: Fluids have certain physical properties, such as density, viscosity, and compressibility, which influence their behavior. Density refers to the mass per unit volume of a fluid, viscosity measures its resistance to flow, and compressibility describes how much the fluid volume changes under pressure.

Bernoulli's Principle: Bernoulli's principle states that within a steady flow of an incompressible fluid, the sum of the pressure, kinetic energy, and potential energy per unit volume remains constant. It describes the relationship between fluid velocity and pressure and is often used to explain phenomena like lift in aerodynamics.

Navier-Stokes Equations: The Navier-Stokes equations are a set of partial differential equations that govern the motion of fluid substances. They describe the conservation of mass, momentum, and energy and are fundamental to solving fluid flow problems.

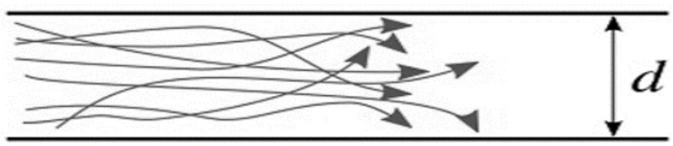
Fluid dynamics finds applications in various fields, including engineering, meteorology, oceanography, aerodynamics, and hydraulic systems. It helps analyze and predict the behavior of fluids in different situations, such as the flow of water through pipes, the movement of air around an airplane wing, or the formation of weather patterns. Computational fluid dynamics (CFD) is a branch of fluid dynamics that utilizes numerical methods and computer simulations to solve complex fluid flow problems.

Streamline flow

Streamline flow refers to the smooth and regular flow of a fluid, where each particle follows a path that is parallel to the direction of the flow (see fig. a). In streamline flow, there is no intermixing of fluid particles between adjacent streamlines. This type of flow is often observed in idealized situations with highly viscous fluids or when the fluid flow is highly laminar and there is minimal turbulence (see fig. b).

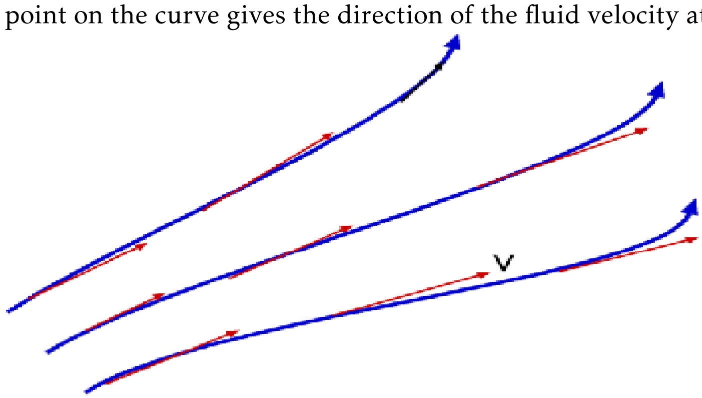
Streamline flow can be visualized by imagining a set of parallel lines (see fig. a) drawn within the fluid, representing the paths followed by individual fluid particles. These lines, known as streamlines, are tangent to the velocity vector of the flow at each point. The streamlines are close together in regions of high velocity and farther apart in regions of low velocity.

(a) Laminar Flow



(b) Turbulent Flow

The path taken by particles of fluid under steady flow conditions are called streamlines. If we represent the flow lines as curves, then the tangent at any point on the curve gives the direction of the fluid velocity at that point.



There is a simple, but very important equation that applies to streamline flow. Suppose in the figure below, that the flow has stabilized so that the amount of fluid that enters area A_1 in time Δt exits from area A_2 in the same time interval Δt.

Then we can write:

$$mass\ going\ into\ A_1 = mass\ going\ out\ of\ A_2 \qquad 1$$

From density equation, we know that

$$m = \rho V \qquad 2$$

Where m is mass, ρ represents fluid density and V represents the volume of the fluid flowing through the area.

Substituting equations (**??**) into (**??**) yields

$$\rho_1 V_1 = \rho_2 V_2 \quad (3)$$

The volume of the fluid through A_1 in time Δt is simply:

$$V_1 = A_1 (v_1 \Delta t) \quad (4)$$

Where v_1 is the average velocity on the cross-section at A_1.

$$\therefore\ V_1 = A_1 (v_1 \Delta t) \ and\ V_2 = A_2 (v_2 \Delta t) \quad (5)$$

Placing equations (**??**) on equations (**??**) yields what is called the continuity equation

$$\rho_1 v_1 A_1 = \rho_2 v_2 A_2 \quad (6)$$

In this form, the equation is restricted to steady streamline flow with no sources (or sinks) for the fluid within the flow region.
A special situation occurs if the fluid is incompressible (such as water), then $\rho_1 = \rho_2$ and we have a relation between the flow rates $v_1 A_1\ and\ v_2 A_2$
$\therefore\ v_1 A_1 = v_2 A_2$ or $v_2 = v_1 \frac{A_1}{A_2}$
In general, the smaller the cross-sectional area, the faster the fluid flow.
Problem 1: A fluid flows steadily through a pipe of varying cross-sectional area. If the velocity of the fluid at one section of the pipe is 5 m/s and the cross-sectional area at that section is 0.2 m², what is the velocity at another section where the cross-sectional area is 0.1 m²?
Solution: According to the principle of continuity, the mass flow rate remains constant in a streamline flow. The mass flow rate is given by the equation:

$$\rho_1 v_1 A_1 = \rho_2 v_2 A_2$$

Since the flow is steady, the mass flow rate is constant. Therefore, we can write:

$$v_2 = v_1 \frac{A_1}{A_2}$$

Plugging in the given values, we get:

$$v_2 = \frac{5\ ms^{-1} \times 0.2\ m^2}{0.1\ m^2}$$

$$v_2 = 10\ ms^{-1}$$

Therefore, the velocity at the section with a cross-sectional area of 0.1 m² is 10 m/s.
Problem 2: Water flows through a horizontal pipe with a velocity of 2 m/s. At a certain point along the pipe, the diameter of the pipe decreases by half. Calculate the velocity of water at this point.

Solution: In this case, we can use the principle of continuity, which states that the product of the velocity and cross-sectional area remains constant in streamline flow. Let's assume the initial diameter of the pipe is D, and after the diameter decreases, it becomes D/2.

According to the principle of continuity, we have:

$$v_1 A_1 = v_2 A_2$$

The cross-sectional area of a pipe is proportional to the square of its diameter. Therefore, we can write:

$$v_1 \times \left(\pi \times (D/2)^2\right) = v_2 \times \left(\pi \times (D/4)^2\right)$$

Simplifying the equation, we get:

$$v_2 = v_1 \times \frac{(D/2)^2}{(D/4)^2}$$

$$\therefore v_2 = v_1 \times 4$$

Plugging in the given velocity (2 m/s), we have:

$$v_2 = 2\ ms^{-1} \times 4 = 8\ ms^{-1}$$

Therefore, the velocity of water at the point where the diameter decreases is 8 m/s.

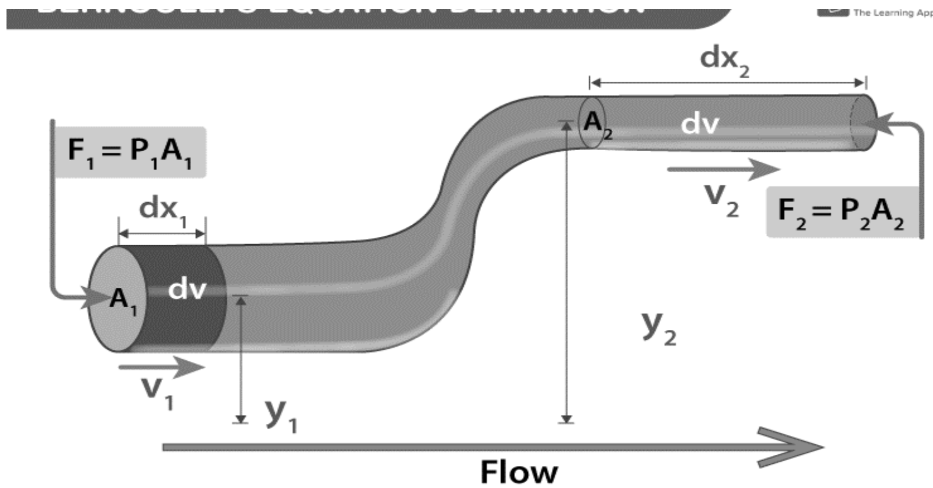
Bernoulli's Principle

The Bernoulli principle, named after the Swiss mathematician Daniel Bernoulli, states that as the speed of a fluid (liquid or gas) increases, its pressure decreases, and vice versa. It is based on the principle of conservation of energy applied to fluid flow.

According to the Bernoulli principle, the total energy of a fluid remains constant along a streamline. The total energy consists of three components: the potential energy (due to the fluid's height or elevation), the kinetic energy (due to the fluid's velocity), and the pressure energy (due to the fluid's pressure).

The qualitative behaviour that is usually labeled with the term "Bernoulli effect" is the lowering of fluid pressure in regions where the flow velocity is increased.

Consider a case of water flowing through a smooth pipe; such a situation is depicted in the figure below. This situation will serve as our working model in obtaining the Bernoulli's equation. We will be employing the work-energy theorem and energy conservation.



If we examine a fluid section of mass m (within the volume dV) traveling to the right as shown in the schematic above. The net work in moving the fluid is:

$$W_{net} = W_1 + W_2 = F_1 dx_1 - F_2 dx_2 \quad (1)$$

Where F denotes a force and x is displacement. The second term picked up a negative sign because the force and displacement are in opposite directions.
From $P = \frac{F}{A}, \, we \, have \, F = PA$ (2)
Substituting equation (**??**) into equation (**??**) we have

$$\Delta W = P_1 A_1 x_1 - P_2 A_2 x_2 \quad (3)$$

The displaced fluid volume V is the cross-sectional area A times the thickness x. this volume remains constant for an incompressible fluid.

$$\therefore V = A_1 x_1 = A_2 x_2 \quad (4)$$

Using eq. (**??**) in (**??**), we have

$$\Delta W = (P_1 - P_2) V \quad (5)$$

Since work has been done, there has been a change in the mechanical energy of the fluid segment.
The energy change between the initial and final position is given by:

$$\Delta E = E_2 - E_1 = (U_2 + K_2) - (U_1 + K_1) \quad (6)$$

Where $U \, and \, K$ are the potential and kinetic energies respectively.

$$\Delta E = \left(mgh_2 + \frac{1}{2} mv_2^2 \right) - \left(mgh_1 + \frac{1}{2} mv_1^2 \right) \quad (7)$$

The work-energy theorem says that the net work done is equal to the change in the system energy. This can be written as:

$$\Delta W = \Delta E \quad 8$$

Substituting eq. (**??**) into (**??**) yields

$$(P_1 - P_2)\,V = \left(mgh_2 + \frac{1}{2}mv_2^2\right) - \left(mgh_1 + \frac{1}{2}mv_1^2\right) \quad 9$$

Dividing eq. (**??**) by the fluid volume V, gives:

$$P_1 - P_2 = \left(\rho gh_2 + \rho\frac{v_2^2}{2}\right) - \left(\rho gh_1 + \rho\frac{v_1^2}{2}\right) \quad 10$$

Where $\rho = \frac{m}{V}$ 11

To complete our derivation, we reorganize eq. (**??**)

$$P_1 = \rho gh_1 + \rho\frac{v_1^2}{2} = P_2 + \rho gh_2 + \rho\frac{v_2^2}{2} \quad 12$$

Finally, we note that eq. (**??**) is true for any two positions,

$$\therefore P + \rho gh + \rho\frac{v^2}{2} = constant \quad 13$$

Where:

1. P is the pressure exerted by the fluid.
2. ρ is the density of the fluid.
3. v is the velocity of the fluid.
4. g is the acceleration due to gravity.
5. h is the height of the fluid above a reference point.

Equation (**??**) is commonly referred to as Bernoulli's equation. This expression is restricted to incompressible fluids and streamline flows.

The Bernoulli equation shows that as the fluid flows faster (increasing velocity), the pressure decreases, assuming the height and density remain constant. This relationship can be observed in various applications, such as airplane wings generating lift, water flowing through a pipe, or air flowing over an object.

It is important to note that the Bernoulli principle is an idealized concept that assumes certain conditions, such as steady flow, incompressibility (for liquids), and low viscosity. In real-world situations, other factors may come into play, such as viscosity, turbulence, and compressibility, which can modify the behavior of fluids.

Top of Form
Problem 1: A pipe has a diameter of 0.2 meters and is carrying water at a velocity of 3 m/s. The pipe is connected to a nozzle with a diameter of 0.05 meters. Find the velocity of the water as it exits the nozzle.
Solution: We can use the Bernoulli equation to solve this problem. The equation states that the sum of the pressure, kinetic energy per unit volume, and potential energy per unit volume of a fluid remains constant along a streamline.
Assuming the pipe and nozzle are at the same height, the potential energy per unit volume cancels out. Therefore, we can write the equation as:

$$P_1 + \frac{1}{2}\rho v_1^2 = P_2 + \frac{1}{2}\rho v_2^2$$

Where: P_1 is the pressure at the pipe (in N/m^2 or Pascal), ρ is the density of water (in kg/m^3), v_1 is the velocity at the pipe (in m/s), P_2 is the pressure at the nozzle (in N/m^2 or Pascal), v_2 is the velocity at the nozzle (in m/s).
Since the pipe and nozzle are connected, the pressure remains the same, so $P_1 = P_2$. Also, the density of water remains constant. Plugging in the values, we have:

$$\frac{1}{2}\rho v_1^2 = \frac{1}{2}\rho v_2^2$$

Canceling out the ρ and solving for v_2, we get:

$$v_2 = \sqrt{\left(v_1^2 \times \left(d_1^2/d_2^2\right)\right)}$$

Where: d_1 is the diameter of the pipe (in meters), d_2 is the diameter of the nozzle (in meters).
Plugging in the given values, we have:

$$v_2 = \sqrt{\left(3^2 \times \left(0.2^2/0.05^2\right)\right)} = \sqrt{(9 \times 6)} = \sqrt{144} = 12\ m/s$$

Therefore, the velocity of the water as it exits the nozzle is 12 m/s.
Problem 2: Water flows through a horizontal pipe with a diameter of 0.1 meters. The pressure at one end of the pipe is 2,000 Pa, and the velocity of the water at that end is 5 m/s. At a certain point along the pipe, the pressure is 1,000 Pa. Find the velocity of the water at that point.
Solution: Using the Bernoulli equation, we can relate the pressures and velocities at the two points along the pipe. The equation is:

$$P_1 + \frac{1}{2}\rho v_1^2 = P_2 + \frac{1}{2}\rho v_2^2$$

Where: P_1 is the pressure at the first point (in N/m^2 or Pascal), ρ is the density of water (in kg/m^3), v_1 is the velocity at the first point (in m/s), P_2 is the pressure at the second point (in N/m^2 or Pascal), v_2 is the velocity at the second point (in m/s).

Plugging in the given values, we have:

$$2000 + \frac{1}{2}\rho\left(5^2\right) = 1000 + \frac{1}{2}(\rho v_2^2)$$

Simplifying the equation, we get:

$$\rho(25) = \rho v_2^2$$

Since the density of water is constant, we can cancel it out. Solving for v_2, we have:

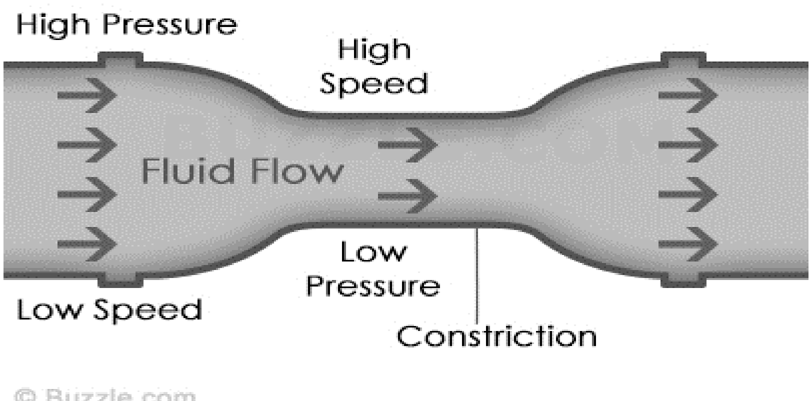
$$v_2 = \sqrt{25} = 5\ m/s$$

Therefore, the velocity of the water at the certain point along the pipe is 5 m/s.

Venturi Effect

The Venturi effect, named after Italian physicist Giovanni Battista Venturi, refers to the phenomenon of fluid flow through a constricted section of a pipe or channel. It describes the relationship between the speed of fluid flow and the pressure exerted by the fluid.

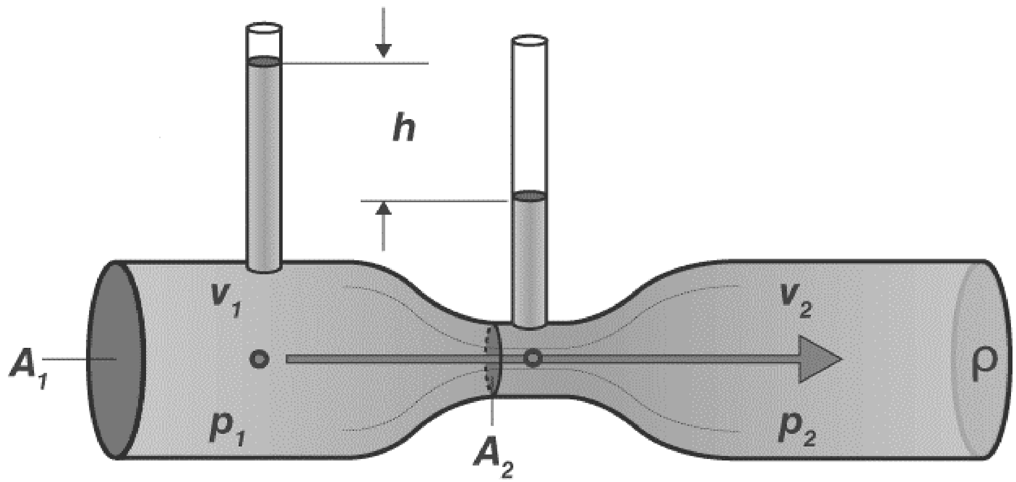
When a fluid (liquid or gas) flows through a pipe, its speed and pressure vary depending on the cross-sectional area of the pipe. The Venturi effect occurs when the fluid encounters a narrow section, called a Venturi, in the pipe.



The Venturi consists of three main parts: an inlet section with a larger diameter, a narrow throat section, and an outlet section with a larger diameter again. As the fluid flows from the wider section into the narrower throat, its speed increases due to the reduction in cross-sectional area. According to the principle of continuity, which states that the mass flow rate remains constant in an incompressible fluid, this increase in velocity is accompanied by a decrease in pressure.

The increase in fluid speed in the Venturi can be explained by the conservation of energy principle. According to Bernoulli's principle, the total energy of a fluid remains constant along a streamline. The total energy consists of three components: potential energy (pressure), kinetic energy (velocity), and gravitational potential energy (height). When the fluid flows into the narrower

throat, the cross-sectional area decreases, resulting in an increase in fluid velocity. As a result, the kinetic energy of the fluid increases while the pressure decreases. This decrease in pressure is sometimes referred to as the Venturi effect.
After passing through the narrowest point of the Venturi, the fluid enters the outlet section, which has a larger cross-sectional area. As the area increases, the fluid slows down, and its kinetic energy is converted back into pressure energy. Therefore, the pressure in the outlet section is higher than in the throat section, but still lower than in the inlet section.



Applying the Bernoulli's equation gives this difference in pressure as:

$$P_1 - P_2 = \frac{\rho}{2}\left(v_2^2 - v_1^2\right)$$

The Venturi effect finds numerous applications in various fields. Some of which include:

1. Carburetors: In internal combustion engines, the Venturi effect is used in carburetors to mix air and fuel. As the air flows through a narrow section, fuel is drawn into the airstream due to the decrease in pressure, creating a combustible mixture.

2. Vacuum cleaners: The Venturi effect is employed in some vacuum cleaner designs to create suction. By forcing air to flow through a narrow section, the pressure decreases, allowing the cleaner to draw in dirt and debris.

3. Respiratory system: The Venturi mask is a medical device used to deliver a controlled oxygen supply to patients. By adjusting the size of the constricted section, the device can control the amount of oxygen mixed with air, thereby providing a specific oxygen concentration.

4. Industrial applications: The Venturi effect is utilized in various industrial processes, such as chemical reactors and fluid measurement devices, where precise control of fluid flow and pressure is required.

Overall, the Venturi effect demonstrates the relationship between fluid flow, velocity, and pressure, showcasing the conservation of energy principles and finding practical applications in numerous fields.

Viscous Flow

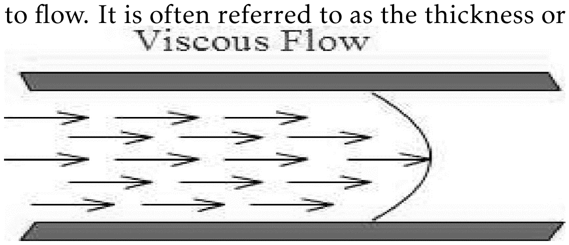
Viscous flow refers to the movement of a fluid that exhibits viscosity, which is the resistance to flow or the internal friction within a fluid. When a fluid flows, its particles or molecules interact with each other and create a shearing effect due to their cohesive forces. Viscosity is a measure of this resistance to shear or deformation.

In a viscous flow, the fluid particles move in layers, with adjacent layers sliding past each other. The velocity of the fluid varies across the flow, with the highest velocity at the center and lower velocities near the boundaries. This phenomenon is known as velocity gradient or velocity profile.

The behavior of a fluid in viscous flow can be described by Newton's law of viscosity, ***which states that the shear stress (τ) between adjacent layers of a fluid is directly proportional to the velocity gradient du/dy,*** where du is the change in velocity and dy is the change in distance perpendicular to the flow direction. Mathematically, this relationship can be expressed as:

$$\tau = \mu \times (du/dy)$$

In this equation, μ is the dynamic viscosity of the fluid, which represents its resistance to shear. The higher the dynamic viscosity, the greater the resistance to flow. It is often referred to as the thickness or "stickiness" of the fluid.



The velocity gradient (du/dy) determines the rate at which adjacent fluid layers slide past each other. When the velocity gradient is high, the fluid experiences a greater shear stress, and the flow is more turbulent. On the other hand, when the velocity gradient is low, the fluid experiences less shear stress, and the flow is more laminar.

If we consider fluid flowing through a pipe; let P_1 be the pressure at point 1 and P_2 that at point 2, a distance L downstream from the point of consideration. The pressure drop becomes:

$$\Delta P = P_1 - P_2$$

This pressure drop is proportional to the flow rate. The proportionality constant is called the resistance R.

$$\therefore \Delta P \propto I_v$$

$$\Rightarrow \Delta P = P_1 - P_2 = I_v R$$

Where $I_v = vA$ is the volume flow rate. The resistance to flow R depends on the length of the pipe L, the radius r, and the viscosity of the fluid.
Viscous flow can be characterized by two different regimes: laminar flow and turbulent flow.

1. Laminar flow: In laminar flow, the fluid moves in smooth, parallel layers, and the velocity profile remains constant over time. The fluid particles flow in an orderly fashion, with minimal mixing and diffusion between layers. Laminar flow occurs at lower velocities and with fluids that have low viscosity. It is described by Poiseuille's law, which relates the flow rate, pressure, viscosity, and dimensions of the system.

2. Turbulent flow: Turbulent flow is characterized by chaotic and irregular fluid motion. The velocity profile fluctuates over time, with the formation of eddies and vortices. Turbulent flow occurs at higher velocities and with fluids that have high viscosity. It is influenced by factors such as Reynolds number, which is a dimensionless parameter indicating the ratio of inertial forces to viscous forces.

The transition from laminar to turbulent flow depends on various factors, including the fluid properties, flow geometry, and velocity. In some cases, the flow may exhibit transitional behavior, which is a mix of laminar and turbulent characteristics.
Viscous flow has significant implications in various fields, including fluid dynamics, engineering, and biology. Understanding the behavior of viscous flows is crucial for designing efficient transportation systems, optimizing industrial processes, analyzing blood flow in the human body, and many other applications.
Problem: As blood flows from aorta through the arteries, arterioles, capillaries, venules, and veins, to the left atrium, the (gauge) pressure drops from about 13.3 kPa to zero. If the flow rate is 0.08 *L/s*, find the total resistance of the circulatory system (*source: Tippler 4th edition*).
Solution: we first convert the pressure from kilopascal to N/m^2

$$13.3\ kPa = 1.33 \times 10^4 N/m^2$$

Using $1L = 1000\ cm^3 = 10^{-3} m^3$

$$R = \frac{\Delta P}{I_v} = \frac{1.33 \times 10^4 N/m^2}{8 \times 10^{-5} m^3/s}$$

$$= 1.66 \times 10^8\ N.s/m^5$$

Poiseuille's law
Poiseuille's law, also known as Hagen-Poiseuille law, describes the flow of a viscous fluid through a cylindrical pipe or a blood vessel. It quantitatively relates the flow rate of the fluid to various factors such as the pressure difference

across the pipe, the viscosity of the fluid, the length of the pipe, and the radius of the pipe. Poiseuille's law is derived from the principles of fluid mechanics and is widely used in various fields, including physiology, engineering, and medicine.

The assumptions of the law are the flow is laminar, viscous and incompressible and flow is through a constant circular cross-section that is substantially longer than its diameter. We are to derive the poiseuille equation using these basic assumptions.

Consider a solid cylinder of fluid, of radius r inside a hollow cylindrical pipe of radius R. The driving force on the cylinder due to the pressure difference is

$$F_{press} = \Delta P\left(\pi r^2\right)$$

The viscous drag force opposing the motion depends on the surface area of the cylinder (length L and radius r).

$$\Rightarrow F_{visc} = -\eta\left(2\pi rL\right)\frac{dv}{dr}$$

In equilibrium condition of constant speed, where the net force goes to zero

$$F_{press} + F_{visc} = 0$$

$$\Rightarrow \Delta P\left(\pi r^2\right) = \eta\left(2\pi rL\right)\frac{dv}{dr}$$

$$\therefore \frac{dv}{dr} = \frac{\Delta P\left(\pi r^2\right)}{\eta\left(2\pi rL\right)} = \frac{\Delta P}{2\eta L}\times r$$

Empirically, the velocity gradient is as shown:

At the center,

$$r = 0$$

$$\Rightarrow \frac{dv}{dr} = 0$$

$\therefore v$ is at maximum at the edge:

$$r = R; v = 0$$

We can integrate $\frac{dv}{dr} = \frac{\Delta P}{2\eta L}\times r$; thus:

$$\int_v^0 dv = \left(\frac{\Delta P}{2\eta L}\right)\times\int_r^R r\,dr$$

$$v\left(r\right) = \left(\frac{\Delta P}{4\eta L}\right)\left[R^2 - r^2\right]$$

The above equation has a parabolic form as expected. From the continuity equation, the volume flow rate is given as:

$$\frac{dV}{dt} = \int v.dA$$

Substituting the velocity profile equation and the surface area of the cylinder, we have:

$$\frac{dV}{dt} = \int v.dA = \int_0^R \left(\frac{\Delta P}{4\eta L}\right)\left[R^2 - r^2\right].(2\pi r dr)$$

$$= \left(\frac{\pi.\Delta P}{2\eta L}\right).\int_0^R \left(R^2 r - r^3\right)dr$$

$$\left(\frac{\pi.\Delta P}{2\eta L}\right)\left[\frac{R^4}{2} - \frac{R^4}{4}\right]$$

$$= \frac{\pi.\Delta P.R^4}{8\eta L}$$

$$\Rightarrow \frac{dV}{dt} = \frac{\pi.\Delta P.R^4}{8\eta L}$$

But $\frac{dV}{dt} = I_v$

$$\Rightarrow I_v = \frac{\pi.\Delta P.R^4}{8\eta L}$$

The pressure drop is then given as

$$\Delta P = \frac{8\eta L}{\pi R^4} \times I_v$$

The equation $I_v = \frac{\pi.\Delta P.R^4}{8\eta L}$ is known as the Poiseuille's equation
Where:

1. I_v represents the volumetric flow rate of the fluid (in cubic meters per second, m^3/s).
2. ΔP is the pressure difference across the two ends of the pipe (in pascals, Pa).
3. r is the radius of the pipe (in meters, m).
4. η is the dynamic viscosity of the fluid (in pascal-seconds, Pa·s).
5. L is the length of the pipe (in meters, m).

Now, let's break down the components of the equation and understand their significance:

1. Volumetric Flow Rate (I_v): The volumetric flow rate represents the volume of fluid passing through a given point in the pipe per unit time. It is expressed in units of volume per unit time, such as cubic meters per second (m^3/s).

2. Pressure Difference (ΔP): The pressure difference refers to the pressure drop across the two ends of the pipe. It determines the driving force for the fluid flow. The greater the pressure difference, the higher the flow rate.

3. Radius of the Pipe (r): The radius of the pipe measures the distance from the center of the pipe to its inner wall. The Poiseuille's law demonstrates that the flow rate is directly proportional to the fourth power of the pipe radius. Thus, a small change in the radius can significantly impact the flow rate. A larger radius allows for higher flow rates.

4. Dynamic Viscosity (η): Viscosity is a measure of a fluid's resistance to flow. It quantifies the internal friction between adjacent layers of fluid as they move relative to each other. Dynamic viscosity represents the ratio of the shear stress to the velocity gradient within the fluid. The higher the viscosity, the slower the fluid flow.

5. Length of the Pipe (L): The length of the pipe indicates the distance over which the fluid flows. The Poiseuille's law shows that the flow rate is inversely proportional to the length of the pipe. A longer pipe will have a lower flow rate compared to a shorter pipe, given the same pressure difference and other factors.

Overall, Poiseuille's law provides an understanding of how different factors influence the flow rate of a viscous fluid through a cylindrical pipe. It demonstrates the importance of parameters such as pressure difference, pipe radius, viscosity, and pipe length in determining the fluid flow.
Top of Form
Problem 1: A fluid with a viscosity of 0.05 Pa·s flows through a pipe of length 2 m and radius 0.02 m. The pressure difference across the ends of the pipe is 100 Pa. Calculate the volumetric flow rate.
Solution: Using the Poiseuille's law equation:

$$I_v = \frac{\pi.\Delta P.R^4}{8\eta L}$$

$$I_v = \frac{\pi \times 100 \times (0.02)^4}{8 \times 0.05 \times 2} = 0.3927 m^3/s$$

Therefore, the volumetric flow rate is approximately 0.3927 cubic meters per second.
Problem 2: Water is flowing through a blood vessel of radius 0.5 mm and length 10 cm. The pressure difference across the ends of the vessel is 2000 Pa. The viscosity of water is 0.001 Pa·s. Find the volumetric flow rate.
Solution: Converting the radius and length to meters: $r = 0.5\ mm = 0.5 \times 10^{-3} m; L = 10\ cm = 10 \times 10^{-2} m$

Using the Poiseuille's law equation:

$$I_v = \frac{\pi.\Delta P.R^4}{8\eta L}$$

$$I_v = \frac{\pi \times 2000 \times \left(0.5 \times 10^{-3}\right)^4}{8 \times 0.001 \times 10 \times 10^{-2}} \approx 0.00393\ m^3/s$$

Therefore, the volumetric flow rate is approximately 0.00393 cubic meters per second.

Reynolds Number

The Reynolds number is a dimensionless quantity used in fluid mechanics to characterize the flow of a fluid (liquid or gas) around a solid object or within a conduit, such as a pipe. It was named after Osborne Reynolds, a 19th-century Irish engineer and physicist who made significant contributions to the study of fluid mechanics.

The Reynolds number (Re) is defined as the ratio of inertial forces to viscous forces within the fluid flow. It is given by the following formula:

$$Re = \frac{inertial\ force}{viscous\ force}$$

The inertial force is given by $\frac{\rho V^2}{L}$; while the viscous force is given by $\frac{\eta v}{L^2}$

$$Re = \frac{\rho V^2}{L} \times \frac{L^2}{\eta v} = \frac{\rho v L}{\eta}$$

Where:

1. Re is the Reynolds number
2. ρ is the density of the fluid
3. v is the velocity of the fluid relative to the object or through the conduit
4. L is a characteristic length of the object or conduit
5. η is the dynamic viscosity of the fluid

The equation can also be written in terms of the radius of the pipe r:

$$Re = \frac{2r\rho v}{\eta}$$

The Reynolds number helps determine the type of flow regime that occurs in a particular situation. It provides information about whether the flow is laminar or turbulent. Laminar flow refers to smooth, ordered flow where the fluid particles move in parallel layers, while turbulent flow is characterized by chaotic, irregular motion with eddies and swirls.

In general, the Reynolds number can be used to make predictions about the behavior of fluid flow. When the Reynolds number is low, below a critical value (around 2,000), the flow tends to be laminar. As the Reynolds number increases, the flow transitions to a turbulent state. The specific value at which this transition occurs depends on various factors, such as the geometry of the object or conduit and the fluid properties.
Understanding the flow regime is crucial because it affects various aspects of fluid dynamics. For example, laminar flow tends to be predictable, with well-defined streamlines and minimal energy losses due to friction. On the other hand, turbulent flow results in increased mixing, higher pressure drop, and greater energy losses. These differences have implications for engineering design, such as in the design of pipes, pumps, and heat exchangers, where the choice of laminar or turbulent flow can impact efficiency and performance.
The Reynolds number is widely used across different fields, including engineering, physics, and biology, to analyze and model fluid flow. It provides a quantitative measure to assess and compare fluid flow situations, enabling researchers and engineers to predict and analyze flow patterns, pressure drops, heat transfer rates, and other relevant characteristics of fluid systems.
Top of Form
roblem: A fluid with a density of 1000 kg/m^3 flows through a pipe at a velocity of 2 m/s. The pipe has a diameter of 0.1 m, and the dynamic viscosity of the fluid is 0.01 kg/(m·s). Determine the Reynolds number and classify the flow regime.
Solution:

1. Identify the given values:
   (a) ρ = 1000 kg/m^3 (density of the fluid)
   (b) v = 2 m/s (velocity of the fluid)
   (c) L = 0.1 m (diameter of the pipe)
   (d) η = 0.01 kg/(m·s) (dynamic viscosity of the fluid)
2. Substitute the values into the Reynolds number formula: $Re = \frac{\rho v L}{\eta} \frac{1000\times 2\times 0.1}{0.01} =$ 2000
3. The Reynolds number is 2000.
4. Determine the flow regime:
   (a) If Re < 2000, the flow is laminar.
   (b) If Re > 4000, the flow is turbulent.
   (c) If 2000 < Re < 4000, the flow is transitional.

In this case, Re = 2000, which falls within the range of 2000 < Re < 4000. Therefore, the flow regime is transitional.

By solving problems involving the Reynolds number, you can analyze fluid flow characteristics and predict the behavior of different systems, such as pipes, channels, or even airflows over objects.
Top of Form
Top of Form

Stoke's Law and Applications

Stokes' law, named after the Irish mathematician and physicist George Gabriel Stokes, describes the behavior of small spherical particles settling in a viscous fluid. It provides a mathematical relationship between the drag force experienced by a particle and its velocity. Stokes' law is applicable in the regime of low Reynolds numbers, which indicates that the flow is laminar and the inertial forces are negligible compared to the viscous forces.

Stokes' law states that the drag force (F) acting on a small spherical particle in a viscous fluid is proportional to the velocity (v) of the particle, the viscosity (η) of the fluid, and the radius (r) of the particle. Mathematically, it can be represented as:

$F = 6\pi\eta rv$

Where: F is the drag force on the particle (in Newtons), η is the viscosity of the fluid (in Pascal-seconds or Poise), r is the radius of the particle (in meters), v is the velocity of the particle relative to the fluid (in meters per second), and π is a mathematical constant (approximately 3.14159).

Stokes' law assumes that the particle is much smaller than the characteristic length scale of the flow, and its motion is dominated by the viscous drag forces. It also assumes that the particle is moving at a constant velocity, without any acceleration.

Applications of Stokes' law in fluid flow include:

1. Sedimentation: Stokes' law is commonly used to estimate the settling velocity of particles in sedimentation processes. It helps in understanding the behavior of solid particles when they settle through a fluid under the influence of gravity. The law enables the calculation of terminal settling velocity, which is useful in various fields such as wastewater treatment, sediment transport studies, and the separation of solid particles from suspensions.

2. Particle Size Analysis: By measuring the settling velocity of particles in a fluid medium, Stokes' law can be used to estimate the particle size. This technique is known as sedimentation analysis or Stokes' settling. It has applications in the characterization of colloidal particles, determination of particle size distributions, and particle separation techniques.

3. Fluid Dynamics: Stokes' law provides insights into the flow behavior of fluids with low Reynolds numbers. It helps in understanding the drag forces acting on small particles, such as microorganisms or nanoparticles, in biological and chemical processes. The law is also relevant in the study of droplet formation, emulsion stability, and the behavior of particles in suspension.

4. Rheology: Rheology is the study of the flow and deformation of materials, particularly non-Newtonian fluids. Stokes' law serves as a basis for understanding the behavior of viscous fluids under shear or extensional flow. While it is a simplification of more complex fluid dynamics, it forms the foundation for some models used in rheological analysis.

5. Microfluidics: In microfluidic systems, where the flow occurs at small length scales, Stokes' law is frequently used to describe the behavior of particles or droplets. By considering the drag forces, researchers can predict the flow behavior and manipulate the motion of particles or droplets in microchannels for various applications, including lab-on-a-chip devices, medical diagnostics, and chemical synthesis.

It is important to note that Stokes' law is an idealization and has certain limitations. It assumes that the fluid flow is steady, the particles are small and spherical, and the flow is laminar with low Reynolds numbers. In real-world situations, deviations from these assumptions may require the consideration of more complex fluid dynamics models.
Top of Form
Problem: A spherical particle of radius 0.1 mm is falling in a fluid with a viscosity of 0.01 Ns/m². The velocity of the particle is 0.5 m/s. Calculate the drag force acting on the particle.
Solution: To solve this problem, we can use Stokes' law equation:
F = 6πηrv
Where: F is the drag force η is the viscosity of the fluid *r* is the radius of the particle *v* is the velocity of the particle
Plugging in the given values:

$$F = 6\pi \times 0.01 \times 0.1 \times 0.5 \approx 0.0942\ N$$

Therefore, the drag force acting on the particle is approximately 0.0942 N.
Top of Form
Top of Form
Top of Form
Top of Form