International Journal of Engineering Research and Reviews ISSN 2348-697X (Online)
Vol. 14, Issue 1, pp: (1-25), Month: January - March 2026, Available at: www.researchpublish.com

Fundamentals of fluid mechanics and its contributions to sustainability of the planet, humans and the economy

Dr Amieyeofori Valentine Felix

PhD, Sustainable Energy Engineering

Entek Integrated Resources Limited, Lagos, Nigeria

DOI: https://doi.org/10.5281/zenodo.18518220

Published Date: 07-February-2026

Abstract: **The subject of fluid mechanics, which deals with both static and dynamics fluids, is the pillar that supports the planetary system, all living things, and also the inanimate global economy as it covers not only the planetary movements, but also handles nearly every facet of life on the planet, such as in breathing, blood flow, swimming, pumps, fans, turbines, airplanes, ships, rivers, windmills, pipes flow, missiles, icebergs, engines, filters, jets, sprinklers, plants, and other living organisms. This paper therefore x-rayed the critical role fluid mechanics plays in supporting human life and the planetary systems from medicine, meteorology, oceanography, hydrology, biofluids, and environmental fluid mechanics all of which deals with naturally occurring flows in human and in the environment. It is also the pivot upon which all forms of transportations are anchored from aerodynamics in aircraft and rockets to hydrodynamics in naval ships and submarines and therefore employs different models of fluid flow patterns. Fluid mechanics is also behind all forms of combustion systems, so also in agriculture and food production that involves irrigation, flood control, water supply, sewage disposal. It is also responsible for global energy and oil and gas operations. Recognizing its vast applications in all aspects of human life, this paper recommends continuous investment in research and development in the study of the various disciplines of fluid mechanics, especially biofluid, environmental and geophysical fluid mechanics. More research is also needed in computational fluid dynamics that can provide concise solutions to the problems of turbulence, since most fluids in life and at industrial scales occur under turbulence regimes.**

Keywords: **Static and Dynamics fluid mechanics, Computational Fluid Dynamics, Flow Patterns, Environmental Fluid Mechanics.**

---

1. INTRODUCTION

Fluid mechanics studies fluid either in motion or at stationary and given that the planet consists of mostly fluid- air, and water, or nearly everything on it moves within or near a fluid, the study of fluid mechanics has significant role in human survival, growth, and development, and covers nearly every facet of life on the planet, such as in breathing, blood flow, swimming, pumps, fans, turbines, airplanes, ships, rivers, windmills, pipes flow, missiles, icebergs, engines, filters, jets, sprinklers, plants, and other living organisms, etc. (White, 2011).

Fluid mechanics according to Cengel and Cimbala (2018) as a branch of mechanics, is the oldest physical science known to man that deals with bodies at rest, called statics, and bodies in motion under the action of forces called dynamics. As a science, fluid mechanics deals with the fluids or fluid statics and fluid in motion or fluid dynamics, plus the interaction of fluids with solids or other fluids at the boundaries. Cengel and Cimbala (2018 added that the fluid at rest, or fluid statics is a special case of motion with zero velocity, thus making the subject of fluid mechanics as synonymous with fluid dynamics. As a study of fluid dynamics, fluid mechanics is subdivided into hydrodynamics, which is the study of incompressible fluids, as in liquids, especially water, and gases at low velocities. While the flow of Liquid is classified as hydraulics, gas flow is referred to as gas dynamics, which is characterized with significant density changes during flow as in the case of gas flow at high pressures through nozzles. A type of gas dynamics is aerodynamics which is concerned with the flow of gases (especially air) over aircraft, rockets, and automobiles at high or low speeds. Fluid mechanics are also studied as meteorology, oceanography, hydrology, biofluids, and environmental fluid mechanics both of which deal with naturally occurring flows in human and in the environment. (Çengel and Cimbala (2018).

White, 2011, stated that all transportation systems operate within one form of fluid medium from aerodynamics in aircraft and rockets to hydrodynamics in naval ships and submarines. Similarly, all combustion systems require one fluid or the other, so also in agriculture and food production that involves irrigation, flood control, water supply, sewage disposal. Fluid flow is also responsible for all projectile motions, and in oil and gas pipelines. Though unknown to them, ancient civilization used the principles of fluid mechanics in sailing ships with oars and also in irrigation systems, while the Romans built extensive aqueduct systems in the fourth century B.C. According to Lumley et al (1996), the practice of fluid mechanics, started as an art in prehistoric times when the Homo sapiens through sheer perseverance, with no knowledge of air resistance or aerodynamics principles, constructed streamlined spears, sickle-shaped boomerangs and fin-stabilized arrows. These technologies were confirmed when archeologists excavated three aerodynamically correct wooden spears in an open-pit coal mine near Hanover, Germany, that dated to about 400,000 years ago.

Historically, fluid mechanics gradually became a subject of interest when Archimedes, the Greek Mathematician in 285-212 BC, discovered the buoyancy principle accidently when he attempted to measure the gold content in the crown of King Hiero II by using nondestructive test. Steadily, the fluid mechanics concepts evolved in application of fluid machinery such as piston pumps, watermill, and windmills. The subject became a science with the formulation of the laws guiding fluid by Sir Isaac Newton (1643–1727) when he explored fluid inertia and resistance, free jets, and viscosity. The works of Daniel Bernoulli (1700–1782), a Swiss, and his associate Leonard Euler (1707–1783) introduced the energy and momentum equations, while Jean Poiseuille (1799–1869) introduced the concept of flow in capillary tubes for multiple fluids. Gotthilf Hagen (1797–1884) differentiated between laminar and turbulent flow in pipes, and gradually the study of fluid mechanics began to attract global interests, from England, with Lord Osborne Reynolds (1842–1912) who developed the dimensionless number that bears his name, and Navier, and George Stokes (1819–1903) that completed the general equation of fluid motion (with friction) that took their names. Ludwig Prandtl (1875–1953), a German, through his pioneering paper in 1904, introduced the boundary layer concept near the walls where the friction effects are significant, and an outer layer with negligible friction effect, but where the simplified Euler and Bernoulli equations are applicable. (Çengel and Cimbala (2018). The mid twentieth century could be referred to as the golden age in fluid mechanics applications, (Çengel and Cimbala (2018), while according to White, 2011, the second half of the twentieth century introduced the Computational Fluid Dynamics (CFD) as a new tool with the advancement in computing in the 1950s, and the emergence of the personal computer in the 1970s, to provide solutions to fluid mechanics problems that could not be easily and readily solved manually. Finally, the late twentieth century according to Çengel and Cimbala (2018, saw aggressive fluid mechanics research especially in the areas of application of high-speed computers to solve complex problems, such as global climate modelling or the optimization of a turbine blade. Srivastava, 2020 contributed to the areas of current research areas in fluid mechanics to include the areas of Turbulence, which is characterized by chaotic and rapidly fluctuating flow fields. Another area of research is in Flow Instability which explores whether a particular flow is stable or not, while High-speed compressible flows, which deals with shockwaves, supersonic and hypersonic flows is also another area of focused research. Srivastava, 2020, also captured the area of Geophysical fluid dynamics, which revolves around atmospheric flows, weather predictions, while the area of Hydrology focuses on water resources, environmental, and subsurface flows. Other current areas of study include Rheology, which deals with constitutive relations for complex fluids; flow of complex (non-Newtonian) fluids, multiphase flows, and around porous media, thin films, particle motion, biological and microscale flows fall. Another area of interest is in magnetohydrodynamics which explains how magnetic fields and fluids interact with each other.

2. THEORETICAL CONCEPTS

Given that the universe is enveloped in one fluid or the other, White, 2011, attributed the subject of fluid mechanics as very crucial to the existence and survival of all living things on the planet. The laws of fluid dynamics, therefore, control a diverse range of activities from how airplanes fly, to how ships sail, how our bodies function, how dust particles settle, as well as how heat is transferred from the sun’s core to the outside, and finally to the formation of magnetic fields around planets. This section thus presents the properties of liquids that are critical to the study of fluid mechanics.

2.1 Properties of Matter.

Since matter exist only two states, either fluid and solid, fluid mechanics study attempts to make a clear distinction on the behaviour of solids, liquids, and gases. Consequently, most engineering fluid mechanics problems focus on separating liquids, such as water, oil, mercury, gasoline, and alcohol, from gases, such as air, helium, hydrogen, and steam, by relying on their temperature and pressure differences. There are, however, some borderlines where some solids such as asphalt slowly deform and exhibit liquid behaviour with increasing temperature, so also some colloids and slurry mixtures, which resist small stress, later begin to flow as fluids under increasing stress. There are also cases where liquids and gases co-exist in two phase mixtures as in steam-water mixtures or as entrapped water in air bubbles. There are yet other cases where liquid and gas become indistinguishable and exist as a single phase more as gas, at temperatures and pressures above the so-called critical point of a substance. The gas with increasing pressure far above the critical point, will gradually begin to behave as liquid and then condenses into liquids. This makes the usual thermodynamic approximations such as the perfect-gas law become inaccurate. (Srivastava, 2020).

Though matter exists in three phases such as solid, liquid and gas, these phases are distinguishable through the application of shear or tangential stress; solids resist shear stress by a static deflection, whereas fluids will initiate motion at the onset of any shear stress applied irrespective of the quantum. A fluid will thus moves and deform continuously under any amount of shear stress, implying that a static fluid is in a state of zero shear stress, a state often called the hydrostatic stress condition in structural analysis. (White, 2011).

Figure 1 shows how a rubber placed in between two parallel plates deforms upon application of a shear force. Generally, in solids, the stress is proportional to strain, while in fluids, the stress is only proportional to the rate of strain. Therefore, solids eventually stop deforming under a constant shear force, whereas a fluid continues to deform and approaches a constant rate of strain.

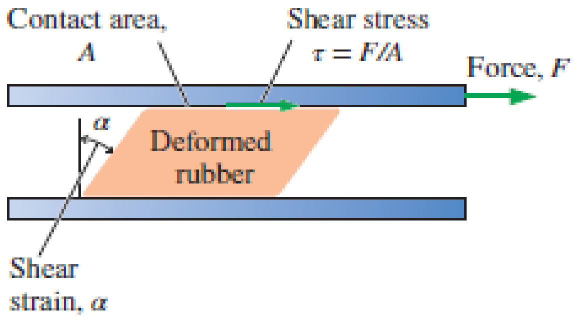


Figure 1: Deformation of a rubber block under shear force. Source: (Çengel and Cimbala (2018).

The rubber solid in between the plates will exhibit an opposing force, F, expressed as $F = \tau A$ at the plate due to friction, under the shear stress, where τ is the shear stress and A is the contact area between the upper plate and the rubber. When the force is removed, the solid rubber automatically is restored to its original state so far as the applied force is within its elastic limit. When a liquid is subjected to the same condition as the solid rubber, with the fluid layer in direct contact with the upper plate, the liquid would continuously move with the velocity of the plate irrespective of the size of the force F. The fluid velocity reaches zero at the lower plate since the fluid velocity is proportional to the depth or distance between the plates due to frictional effect. Liquids generally takes the shape of its container with a free surface in a gravitational field due to its closely molecular bonds, while gases, expand until they encounter the walls of the container and fills the entire available space due to its loose bonding, and therefore does not form any free surface in the container as shown in figure 2.

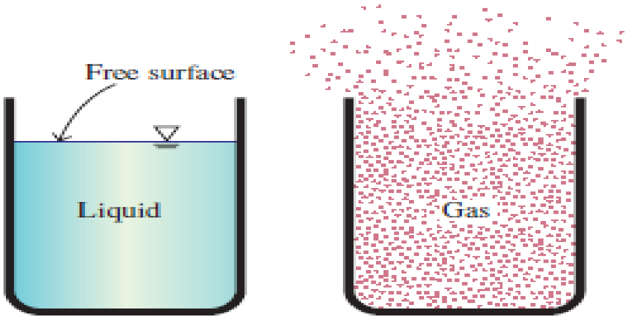


Figure 2. The molecular behaviour of liquids and gas in a container. Source, (Çengel and Cimbala (2018).

Figure 3 shows the different intermolecular bonds structures of solids, liquids and gasses , with solids, having the strongest bonds, while gas has the weakest bonds. This is because the molecules in solids are more closely packed together, whereas those in gases are loosely separated far apart. Molecules in liquids move about each other

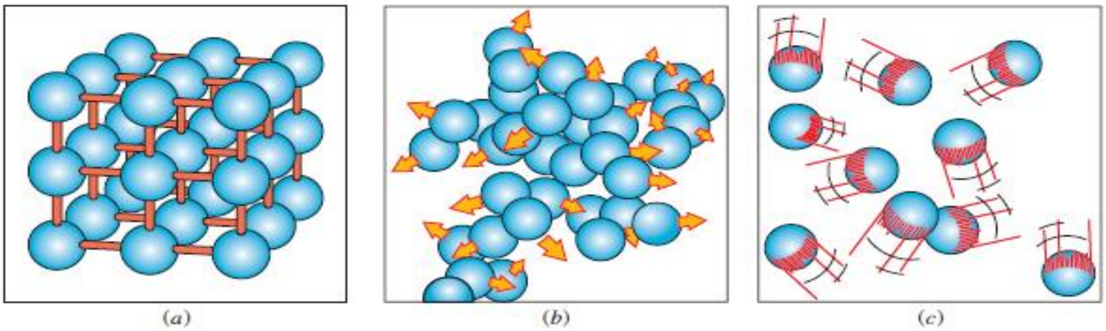


Figure 3. The arrangement of atoms in different phases: (*a*) solids, (*b*) liquids phase, and

(*c*) gas phase. Source: (Çengel and Cimbala (2018).

Gas molecules exhibit higher energy levels than liquids and solids, making it rather difficult to condense unless it releases a large amount of this energy. (Çengel and Cimbala (2018).

2.2 The Fluid Scales

Srivastava, 2020 showed that fluids exist at different scales, and that to fully appreciate the role of fluid mechanics, it is important to understand the predominant fluid scale, as fluid behaves differently at different scale levels. This is because, the physical dimension of flow is an important quantity in the study of fluid mechanics. The first level is the nanometer scale, or 10^{-9} meters, while there is also an astrophysical flow in the range of 10^{9} meters. However, one amazing feature of fluid dynamics is that, regardless of the scale, the same principles, laws (conservation of mass, momentum, and energy), and even equations are applicable at all levels. The next set of scales are the atomic and molecular levels, where the discrete nature of matter becomes prominent and the continuum hypothesis breaks down, and where the fluid no longer remains as fluid and the laws of fluid dynamics are no longer applicable at this scale. The micro-level (1 to 100 microns) fluid scales are called micro-fluids, and deal with flows in small chips, around bacteria, porous media, dust particles, etc. The next scale level are the millimeter-scale dimensions, ranging from several millimeters to a maximum of a few meters, which consist of flows in engineering devices such as in small pipes, arteries, and veins, etc. On top of this scale are those flows in few tens of meters such as flows around big objects like buildings, cars, airplanes, etc, while we have flows in the scales of thousands of kilometers in cases such as flows in or around the atmosphere, oceans, landmarks, etc. Lastly, and which is the highest scale is the astrophysical flows which includes flows in stars and gas clouds.

2.3 Classification of Fluid Flows

There are different classifications of fluid flow that are of interest in the study of fluid mechanics, and this section presents a set of these classifications. However, prior to delving into the different classifications of fluid flow, we present the concept of the continuum hypothesis in fluid flow. Though fluid flows in a discontinuous or discrete manner at microscopic scales, the subject of fluid mechanics is not really concerned with the discrete molecular structure of matter, but rather in the gross behaviour or the average manifestation of the molecular motion of the fluids, which in principle gave rise to the continuum hypothesis. The continuum or macroscopic approach is based on the size of the flow system, which is determined by the size of the body around which the flow is taking place. Technically, the size of flow must be bigger than the mean free path of the molecules, which is the case most of the time since the mean free path is usually very small. This continuum hypothesis breaks down at the upper altitudes of the atmosphere, as the mean free path of the molecules in those regions are of the order of a meter, making it impossible to apply the continuum hypothesis. In its place, the kinetic theory becomes the tool for the study the dynamics of these rarefied atmospheric gases. (Kundu and Cohen, 2008).

Developing from this macroscopic or continuum model, the study of fluid flow can be grouped based on the behaviour of fluid under different conditions, and for ease of study, this section presents some of the basic fluid flow classification. Fluid flow according to Bright Hub Engineering (2009) can be based on variation with time, and variation with space. The variation of the fluid flow parameters with time produces steady or unsteady flow, while the variation with space produces either uniform or non-uniform flow.

The section below thus discusses the various flow classification.

2.3.1 Viscous versus Inviscid Regions of Flow

Viscosity which is the fluid property that causes internal resistance to flow, is itself the product of the cohesive forces between liquid molecules and by the molecular collisions in gases. It thus measures the internal stickiness of the fluid. Technically, all fluids exhibit one form of viscous flow, as there is no fluid with zero viscosity. However, in some real time flow, there exist some regions, typically not close to solid surface that exhibit negligible viscous forces compared to inertial or pressure forces. These flow regimes are called Inviscid Flow, which is a simplification of solving fluid mechanics problems is as shown in figure 4. The viscous and the inviscid regions in figure 4 developed when a flat plate parallel was inserted into the fluid stream with uniform velocity.

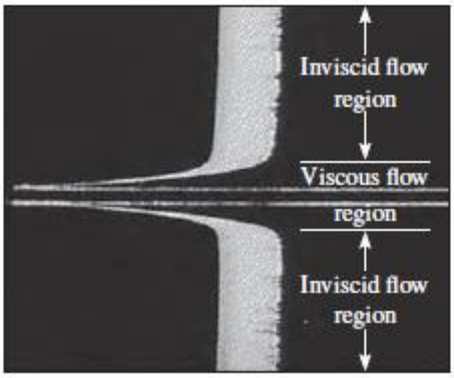


Figure 4: Diagramtic representation of the viscous and inviscid flow region. Source: (Çengel and Cimbala (2018).

Due to the viscous force, and no slip condition, the fluid adheres to both sides of the plate in order to create the viscous flow region, while the region away from the plate that are unaffected by the presence of the plate exhibit the inviscid flow region.

2.3.2 Internal versus External Flow

Technically, a flow is said to be internal such as in pipeflow, when the flow is bounded by solid surfaces, while it is called external flow where the flow is unbounded over a surface such as a plate. Open channel flow is liquid flow in a duct that is partially filled with a free surface, such as in the flow of water in rivers and irrigation ditches. It should be noted that internal flows are influenced by viscosity throughout the flow, while the viscous effect in external flows are limited only the boundary layers near the solid surface, and regions downstream of the bodies.

2.3.3 Compressible versus Incompressible Flow

A compressible flow is characterized by significant changes in fluid density when the fluid moves with very high velocity similar to its sonic velocity, and this is only possible with gas flows, since very high pressures in the range of 1000 atms are required to initiate sonic velocity in liquids. Gases, however, can readily exhibit sonic flow at pressure ratio of only 2:1. (White, 2011). Incompressibility on the other hand is an approximation, where the fluid density remains nearly constant throughout, making the fluid volume to remain unchanged at any time in the entire flow period. These are the key characteristics of liquids with constant densities and reasons why liquids are termed incompressible substances. For instance, a pressure of 210 atm, on liquid water only causes 1 atm change in density, while a pressure of 0.01 atm on gas causes as high as 1 percent change in density. Gas can also be classified as incompressible where its density changes are below 5 percent, with Mach number of less than 0.3. At this state, air compressibility can therefore be neglected at room temperature at speed less than 100 m/s. (Çengel and Cimbala (2018).

2.3.4 Laminar versus Turbulent Flow

Laminar flow exhibit ordered fluid motion characterized by smooth layers, such as in the flow of high viscosity oil at low velocities. Turbulent flow occurs at high velocities and are characterized by highly disordered fluid motion with velocity fluctuations. Flow such as low viscosity air at high velocities exhibits turbulent characteristics. Transitional flow alternates between laminar and turbulent, as shown in figure 5.

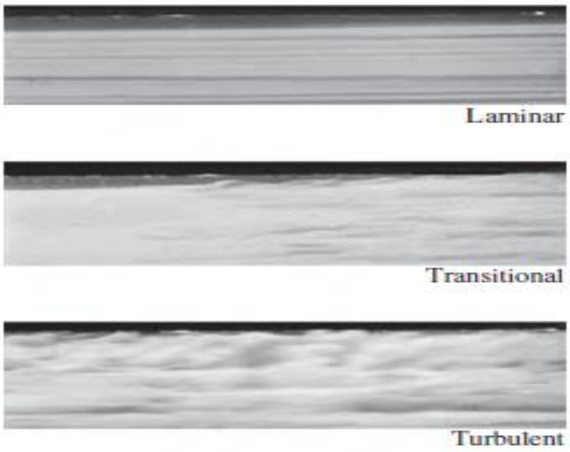


Figure 5. Laminar, Transitional and Turbulent flow characteristics. Source: (Çengel and Cimbala (2018).

2.3.5 Natural (or Unforced) versus Forced Flow

A fluid is said to be in forced flow, when the flow is induced by an external force such as pumps or fans, while natural flow occurs due to natural means such as buoyancy effects caused by the alternating rise and fall in warmer and cooler conditions.

2.3.6. Steady versus Unsteady Flow

Steady and Unsteady flow according to Bright Hub Engineering (2009) are classified based on time variation with the basic flow parameters. The flow is said to be steady flow its velocity, pressure, density, and discharge do not vary with time or are independent of time, while unsteady flow is characterized with flow parameters variation with time. Steady flow rarely occur in real conditions, since flow parameters vary with time, though at very small range such that the average parameter appears constant at some instantaneous time. Steady flow therefore does not mean constant velocity and acceleration as in flow in a curved pipe or through a nozzle. (Ngo, and Gramoll, ()). Other equipment such as turbines, compressors, boilers, condensers, and heat exchangers are also classified as steady-flow devices since this equipment run for much longer periods with little flow parameters variations. It should also be noted that fluid properties under steady flow may remain constant at a particular fixed point but change from point to point within the device. However, since steady-flow devices operate for longer period of time, the fluid properties such as volume, mass, and the total energy content appear to remain constant in steady operation. On the contrary, cyclic devices, such as reciprocating engines or compressors, do not exhibit the steady-flow conditions due to the pulsating tendencies of both the inlet and outlet flow conditions. This now leads to unsteady flow, which is flow characterized by large, alternating, swirling, turbulent eddies, with its fluid properties heavily dependent on time (i.e., $T = T(x, y, z, t)$, $p = p(x, y, z, t)$ and $\rho = \rho(x, y, z, t)$). Unsteady flows is further categorized as periodic flow, nonperiodic, and random flow as shown in figure 7 (Ngo, and Gramoll, ()).

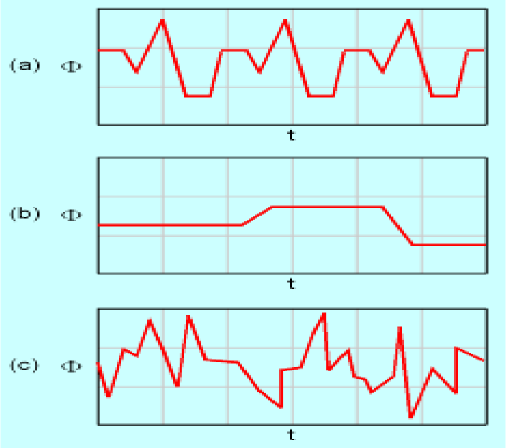


Figure 7: Types of Unsteady Flow: Periodic flow (a), Non-Periodic Flow (b) Random flow (c). Source: (Ngo, and Gramoll, ()).

2.3.7. Uniform versus Nonuniform Flow

Uniform and non-uniform flow falls under flow classification with variation in space or distance. (Bright Hub Engineering, 2009). A flow is said to be uniform when its parameters remain constant throughout the flow path, while non uniform flow changes in its parameters at different points on the flow path. One condition for uniform flow is that the cross-sectional area of the flow path must remain constant, as in case with pipe flow of uniform diameter: flow through a pipeline with variable diameter would be non-uniform. (Bright Hub Engineering (2009). Figure 8 thus depicts uniform flow just downstream of a well-rounded pipe entrance, except for a very thin boundary layer near the wall. Typically, flow in ducts and pipes and at inlets and outlets are approximated as uniform flow in engineering practice for simplicity in calculations. So, while the fully developed pipe flow velocity profile of Fig. 8 is certainly not uniform, for ease of calculations, uniform flow has to be adopted. While this approximation simplifies the calculation, it in turn introduces some error terms in process that requires correction factor.

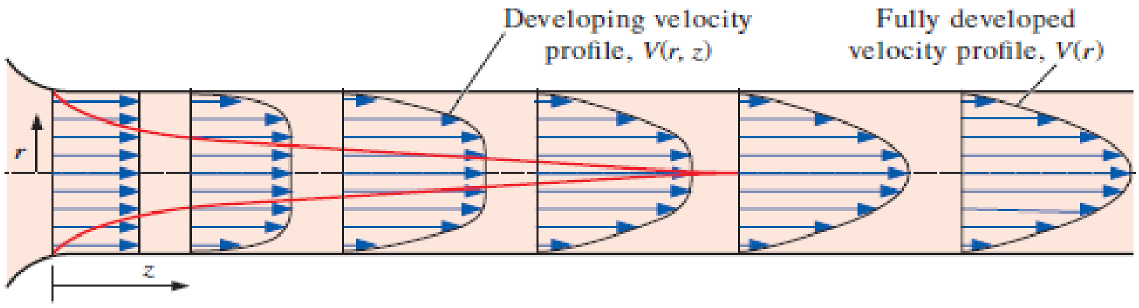


Figure 8: The development of velocity profile in a circular pipe. Source: (Çengel and Cimbala (2018).

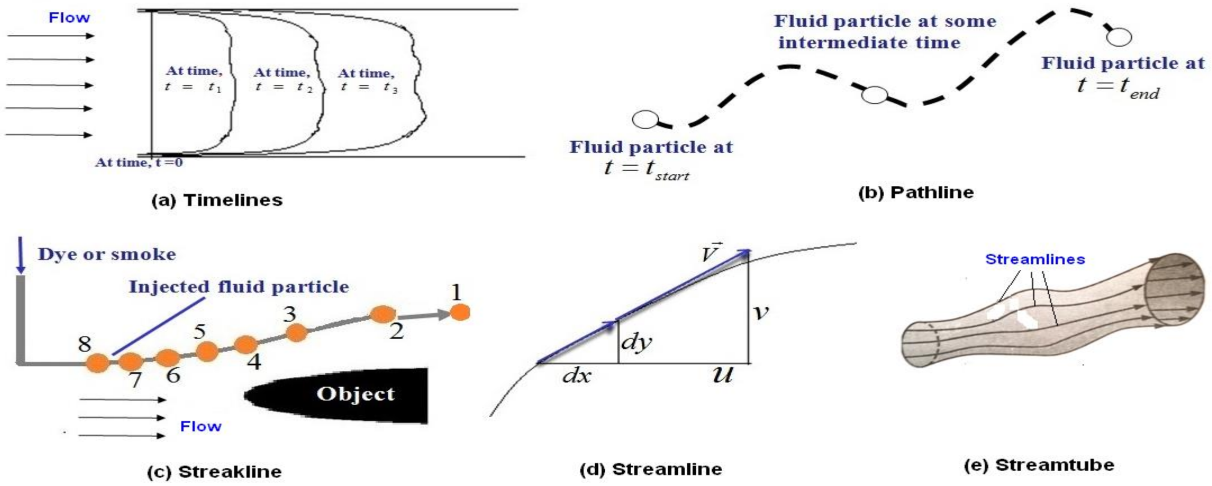
Technically, a flow can be steady and either uniform or non-uniform, and unsteady and be uniform or non-uniform. Steady and uniform flow occurs in pipes with constant diameter and a constant rate of discharge, while a steady and non-uniform flow occurs in a tapering pipe with a fixed discharge rate, such as in water flow through a river with a constant discharge. An unsteady and uniform flow occurs through a pipe with constant cross section and sudden changes in fluid discharge or pressure, while an unsteady and non-uniform flow is characterized by a pressure surge in a flow through a pipe of variable cross section, as in water flow in the network of canals during water release. (Bright Hub Engineering, 2009).

2.3.8 One-, Two-, and Three-Dimensional Flows

A flow is said to be one-, two-, or three-dimensional based on the directions of its flow velocity whether in one, two, or three primary dimensions. A three-dimensional flow has its velocity varying in all three dimensions usually rendered either in cartesian or rectangular coordinates as [$V›$ (x, y, z) or as $V›$ (r, θ, z) in cylindrical coordinates. A flow can, however, be modelled as either one- or two-dimensional flow if its variation in the third direction is relatively small compared to the variations in other directions. With reference to figure 8, due to no-slip condition, the flow at the entrance region of the pipe is two-dimensional since fluid velocity changes in only in the r- and z-directions, and not in the θ-direction. This velocity profile as shown in figure 8, remains unchanged even at some distance from the inlet, approximately about 10 pipe diameters in turbulent flow, as it develops fully, making the flow in this region to be fully developed. It should be noted that whether a flow is one dimensional or two or three also depends on the coordinate system and its orientation, for instance, the pipe flow in figure 8 is one-dimensional in cylindrical coordinates, but two-dimensional in Cartesian coordinates. Again, a flow may be described as two-dimensional if its aspect ratio- (diameter/length) is large such that the flow does remains constant along the longer dimension, as seen in the flow of air over a car antenna with a uniform airflow around the region of the antenna, since the antenna’s length is much greater than its diameter. (Çengel and Cimbala (2018).

2.4 Flow Patterns

Given the highly visual nature of fluid mechanics, fluid flow has been reduced to visible patterns, and there are four basic types of these visible flow patterns, which are Streamlines, Streaklines, Pathlines and Timelines. Figure 9 shows the diagrammatic representation of the various flow patterns in fluid mechanics.



(a) Timelines (b) Pathline (c) Streakline (d) Streamline (e) Streamtube

Figure 9: Diagramatic representation of the various flow patterns: Source. Ray Solutions (2018)

This section presents brief description of each of these flow patterns that we encounter in the study of fluid mechanics.

2.4.1 Streamlines

A streamline is a tangential line to the instantaneous velocity vector. Streamlines do not cross since the velocity at any point in the flow has a single value, except at the stagnation point where the velocity magnitude is zero. A collection of streamlines into a bundle is called Streamtubes as shown in figure 10. Technically, the mass flow rate along a streamtube is constant under steady, one-dimensional flow. Also the cross-sectional area of the streamtube provides information on the local velocity, under a constant density flow. A streamline can be visualized when a drop of water is marked with fluorescent dye and illuminated with laser. As the drop moves under the local velocity fluid, short pictures are taken to show the path. A streamline is formed when several drops are marked and their pictures taken as the move under the velocity field.. (efluids, 2021)

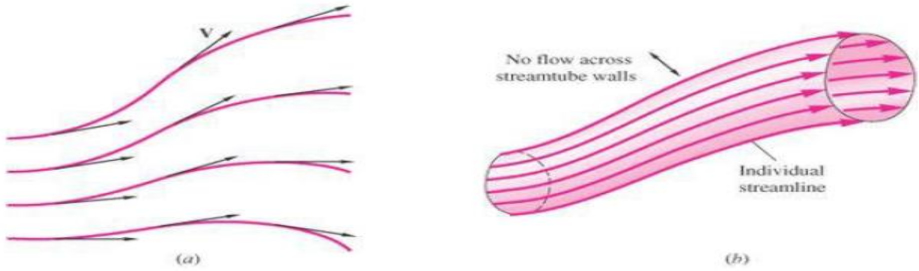


Figure 10: Streamlines (a) and Streamtubes (b). Source: efluids, 2021

2.4.2 Pathlines

A Pathline describes the trajectory of an individual fluid particle in an unsteady flow field, as shown in the figure 11. Pathlines are like streamlines except that pathlines are found in unsteady flow field. It indicates the direction of the particle velocity at successive instants in the flow path. A pathline is thus formed when a particle placed in a flowing water intuitively flows the flow direction at each time as the water flow changes over time or undergoes unsteady flow. the ball intuitively will follow the flow direction at each time step.

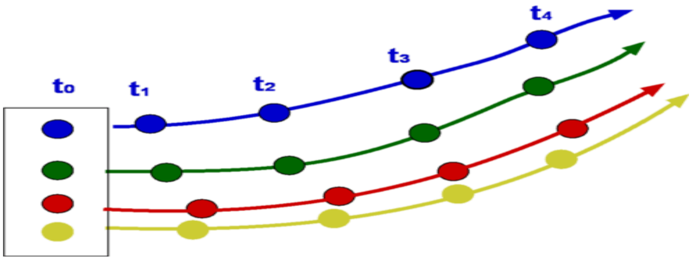


Figure 11: Diagramatic presentation of a Pathline. Source: efluids, 2021

2.4.3 Streaklines.

A streakline is an integrated flow pattern that describes the locus of points of all the fluid particles that have passed continuously through a particular spatial point in the past. It is created by placing multiple particles into the water flow at the same spot but at different time steps. The streakline is the path by connecting all the balls in the placement order. Streaklines are generally closely related to pathlines. A streakline under steady flow, where there is no change in the flow pattern, is the same as a streamline and the path line of a particle, whereas under unsteady flow, a streak line at an instant is the locus of end points of particle paths (or path lines) that started at the instant the particle passed through the injection point.

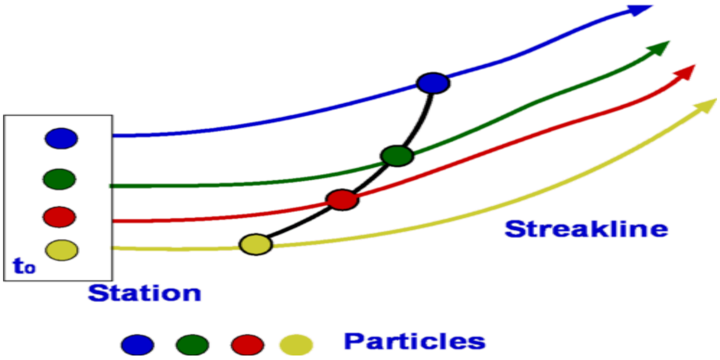


Figure 12: Representation of a Streakline. Source: efluids, 2021

2.4.4. Timeline

A timeline describes the pattern of a curve formed by a set of fluid particles marked at a previous time, which is constantly displaced over successive time as the particles move from one position to another as shown in figure 13. It is created when several particles, small balls are placed into a flowing water stream and allowed to follow the flow direction. A timeline is also an integrated flow pattern since the flow pattern continually distorts with time. (Aerospace, Mechanical & Mechatronic Engg ,2005)

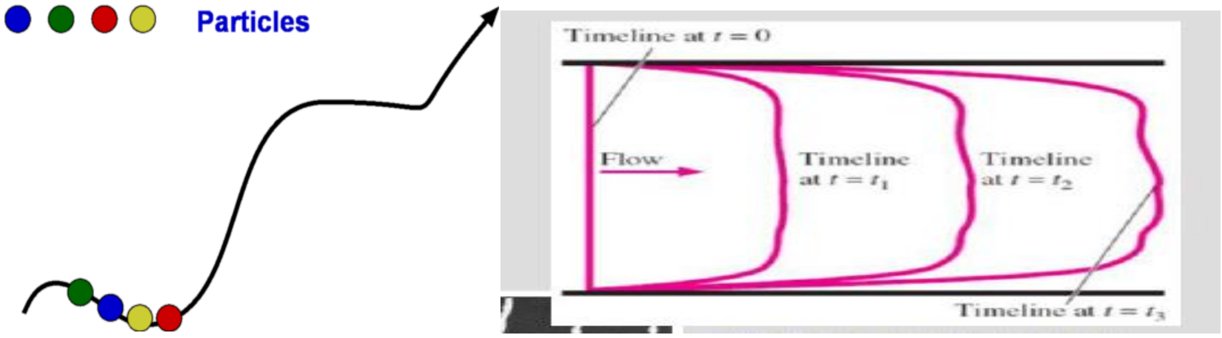


Figure 13: Diagramatic representation of Timelines. Source: Cengel and Cimbala (2010)

It should be noted that under steady flow, streamlines, streaklines, and pathlines are all identical, while they are quite different under unsteady flow. According to White, 2011, Streamlines are convenient to approximate mathematically, while the others can only be approximated experimentally. Also, Streamlines are the most common mathematically to visualize. Again, because the fluid within a streamtube is confined, there is no cross flow through a Streamtube.

2.5 Basic Equations of Fluid Mechanics

This section presents the basic equations that guides the solution path in fluid mechanics, all of which are derived from the conservation equation, and these are the continuity equation or conservation of mass, the momentum equation or the conservation of linear momentum, and the energy equation or the conservation of energy. All three equations are based on the Eulerian (or control volume) model of solving the velocity problems in dynamic flow as illustrated in figure 14, which is a flow draining from a sink. (Janna, (2010)

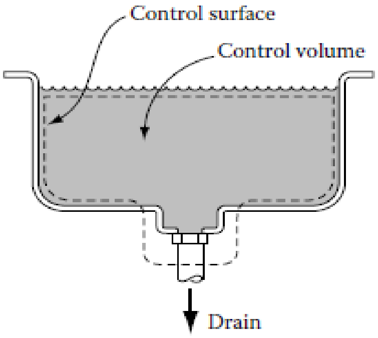


Figure 14: Illustration of Control Volume. Source: Janna, (2010)

The control volume is contained within the dashed line called the control surface and depends on the type of fluid mechanics problems. Also, the control surface is dynamic and can therefore move or change its shape with time.

Relying on the Eulerian model of control volume, a general conservation equation is obtained, by defining N as a flow quantity (mass, momentum, or energy), associated with a fluid volume, and n as the flow quantity per unit mass. Also, the assumption is that two time periods are involved, and through several substitutions, we arrive at the form (Janna, (2010):

$$\underbrace{\left.\frac{dN}{dt}\right|_{S}}_{\text{total particles}} = \underbrace{\left.\frac{\partial N}{\partial t}\right|_{CV}}_{\text{amount stored}} + \underbrace{\iint\limits_{CS} n\rho V_n \, dA}_{\substack{\text{net rate out} \\ \text{(out minus in)}}} \qquad \text{Equation 1}$$

where s = system, cv = control volume, and cs = control surface

From this general conservation equation, we can now derive the various equations. The continuity equation is derived with N, the flow quantity now becoming m, and n becomes 1; m/m, and which when substituted into the general conservation equation, becomes:

$$\left.\frac{dm}{dt}\right|_{S} = \left.\frac{\partial m}{\partial t}\right|_{CV} + \iint\limits_{CS} \rho V_n \, dA \qquad \text{Equation 2}$$

Expressing this in average velocity, translates the continuity equation to become

$$\sum_{\text{in}} \rho AV = \sum_{\text{out}} \rho AV + \left.\frac{\partial m}{\partial t}\right|_{CV} \qquad \text{Equation 3}$$

The continuity equation, according to Janna, 2010, is a conservation of mass equation, that is used to mainly to account for transfer of mass across boundaries and mass storage within control volumes.

Following this, is the conservation of momentum equation, which is also derived from the general conservation equation, either in the form of linear momentum or angular momentum.

Linear momentum is derived by substituting N, the flow quantity to momentum mV, and n, which is N per unit mass, with V, velocity:

$$\left.\frac{d}{dt}(mV)\right|_{S} = \left.\frac{\partial}{\partial t}(mV)\right|_{CV} + \iint\limits_{CS} V\rho V_n \, dA \qquad \text{Equation 4}$$

This equation is further simplified using Newton law of linear momentum to becomes:

$$\sum F = \frac{d}{dt}(mV)\Big|_{S} = \frac{\partial}{\partial t}(mV)\Big|_{CV} + \iint_{CS} V(\rho V \cdot dA)$$

Equation 5

Where, according to Newton law, ΣF is the sum of all forces applied externally to the control volume, such as forces due to gravity, electric and magnetic fields, surface tension effects, pressure forces, and viscous forces (friction). The rate of storage of linear momentum is represented by the first term on the right-hand side, while the last term represents the net rate out (out minus in) of linear momentum. The Linear momentum equation is reduced to the form in equation 6 under a steady one-dimensional flow:

$$\sum F_i = \iint_{CS} V_i \rho V_n \, dA$$

Equation 6

The angular momentum equation is relevant in evaluating moments exerted by moving fluid volumes such as in rotating machinery, viz turbines and pumps. In deriving the angular momentum equation from the linear momentum equation, the fluid is passed in a xy plane with a distance, r from the origin, through a control volume as shown in figure 15 (Janna, (2010).

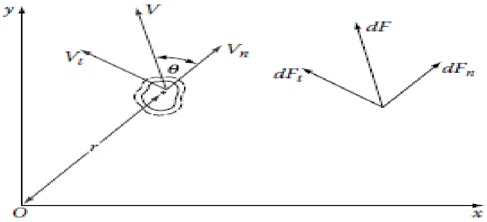


Figure 15: Diagrammatic representation for derivation of the angular momentum equation.

The force on the fluid is resolved into two components, a normal and a tangential force to r. The tangential component is what is adopted in deriving the angular momentum equation:

$$dF_t = \frac{\partial}{\partial t}(V_t \, dm) + (V_t \rho V_n \, dA)$$

Equation 7

The differential torque is then

$$dT_0 = r\frac{\partial}{\partial t}(V_t \, dm) + r(V_t \rho V_n \, dA)$$

Equation 8

Upon integration, and $V\sin\theta = V_t$ and rewriting in vector form, gives the form of the angular momentum equation as:

$$T_0 = \frac{\partial}{\partial t}\iiint_{CV} (\mathbf{r} \times \mathbf{V})dm + \iint_{CS} (\mathbf{r} \times \mathbf{V})(\rho \mathbf{V} \cdot d\mathbf{A})$$

Equation 9

The energy equation is derived from the conservation equation by adopting the law of conservation of energy which states that energy is conserved and not destroyed as it moves from one place to another, and this is expressed as (Janna, (2010):

$$E_2 - E_1 = \bar{Q} - W'$$

Equation 10

or

$$dE = d\bar{Q} - dW'$$

Equation 11

By substituting N =E and n =e in the general conservation equation, the energy equation becomes:

$$\left.\frac{dE}{dt}\right|_{S} = \left.\frac{\partial E}{\partial t}\right|_{CV} + \iint_{CS} e\rho V_n \, dA \qquad \text{Equation 12}$$

Which further becomes

$$\left.\frac{dE}{dt}\right|_{S} = \frac{d\bar{Q}}{dt} - \frac{dW'}{dt} = \left.\frac{\partial E}{\partial t}\right|_{CV} + \iint_{CS} e\rho V_n \, dA \qquad \text{Equation 13}$$

Or

$$\frac{d(\bar{Q} - W')}{dt} = \left.\frac{\partial E}{\partial t}\right|_{CV} + \iint_{CS} \left(u + \frac{V^2}{2} + gz\right)\rho V_n \, dA \qquad \text{Equation 14}$$

where W^I consists of all forms of work, shaft work, electric and magnetic work, viscous shear work, and flow work, crossing the boundary. Developing an expression for dW^I/dt, and substituting for Enthalpy, the conservation of energy equation becomes:

$$\left.\frac{d}{dt}(\bar{Q} - W)\right|_{S} = \left.\frac{\partial E}{\partial t}\right|_{CV} + \iint_{CS} \left(h + \frac{V^2}{2} + gz\right)\rho V_n \, dA \qquad \text{Equation 15}$$

And for the special case of steady, one-dimensional flow, the energy equation becomes:

$$\left.\frac{d}{dt}(\bar{Q} - W)\right|_{S} = \left[\left.\left(h + \frac{V^2}{2} + gz\right)\right|_{out} - \left.\left(h + \frac{V^2}{2} + gz\right)\right|_{in}\right]\rho A V \qquad \text{Equation 16}$$

Another basic equation is the Bernoulli equation, which brings together pressure, velocity, and position or elevation in a flow field. The Bernoulli's equation is derived from the momentum equation under a condition of streamline, steady and frictionless flow, with no viscous effects, but only pressure and gravity effects. Figure 16 is an illustration of the derivation of the Bernoulli's equation (Janna, (2010)

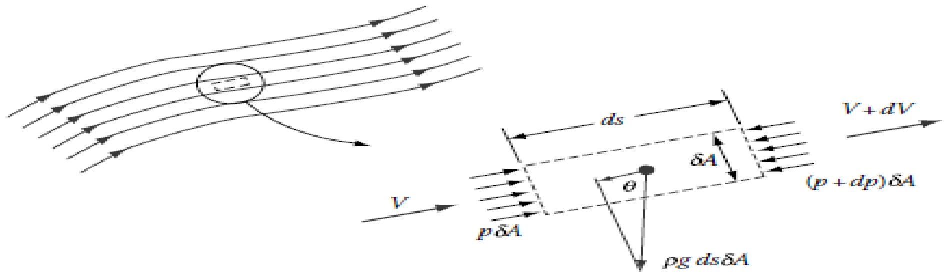


Figure 16. A differential control volume for the derivation of Bernoulli's equation.

The momentum equation is further integrated along the streamtube in the s-direction of the control volume to become:

$$\sum F_s = p\delta A - (p + dp)\delta A - \rho g \; ds \; \delta A \; \cos\theta \qquad \text{Equation 17}$$

With further simplification by assuming an incompressible fluid, and constant density, the momentum equation transforms into the Bernoulli equation as:

$$\frac{p}{\rho}+\frac{V^2}{2}+gz=\text{a constant}$$

Equation 18

Another equation that is pivotal in solving fluid mechanics problem, according to (Janna, (2010) is the Navier Strokes equation, which is the expression of the momentum or continuity equation in differential form in each of the three coordinate directions as shown in figure 17. The assumption is that the fluid is Newtonian with constant properties of density and viscosity

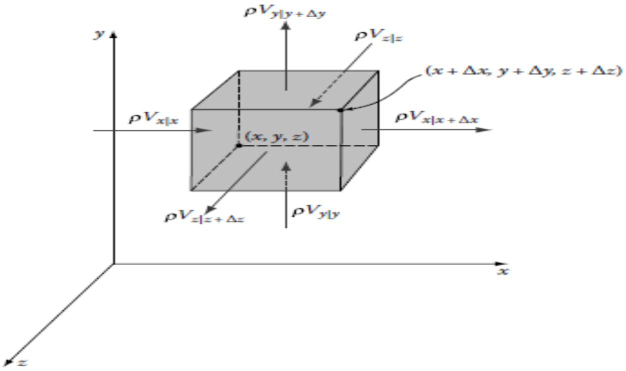


Figure 17: Diagrammatic representation of the derivation of the Navier Strokes Equation. Source: (Janna, (2010)

So, considering the continuity equation

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho V_x)}{\partial x}+\frac{\partial(\rho V_y)}{\partial y}+\frac{\partial(\rho V_z)}{\partial z}=0$$

Equation 19

The equivalent Navier Strokes equations in each direction is given as

x-component:

$$\rho\left(\frac{\partial V_x}{\partial t}+V_x\frac{\partial V_x}{\partial x}+V_y\frac{\partial V_x}{\partial y}+V_z\frac{\partial V_x}{\partial z}\right)$$
$$=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 V_x}{\partial x^2}+\frac{\partial^2 V_x}{\partial y^2}+\frac{\partial^2 V_x}{\partial z^2}\right)+\rho g_x$$

Equation 20

y-component:

$$\rho\left(\frac{\partial V_y}{\partial t}+V_x\frac{\partial V_y}{\partial x}+V_y\frac{\partial V_y}{\partial y}+V_z\frac{\partial V_y}{\partial z}\right)$$
$$=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 V_y}{\partial x^2}+\frac{\partial^2 V_y}{\partial y^2}+\frac{\partial^2 V_y}{\partial z^2}\right)+\rho g_y$$

Equation 21

z-component:

$$\rho\left(\frac{\partial V_z}{\partial t}+V_x\frac{\partial V_z}{\partial x}+V_y\frac{\partial V_z}{\partial y}+V_z\frac{\partial V_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 V_z}{\partial x^2}+\frac{\partial^2 V_z}{\partial y^2}+\frac{\partial^2 V_z}{\partial z^2}\right)+\rho g_z$$

Equation 22

However, according to Cengel and Cimbala, 2018, since these equations are applicable in all directions within the flow field, they are therefore particularly useful in solving flow problems in all flow domains, but manually, and with the use of computer as most differential equations encountered in fluid mechanics are difficult to solve except with the aid of a computer.

The Navier-Stokes equation can also be expressed in vector form by combining all three equations, to give the notable **Navier–Stokes equation** for incompressible flow with constant viscosity:

$$\rho\frac{D\vec{V}}{Dt}=-\vec{\nabla}P+\rho\vec{g}+\mu\nabla^2\vec{V}$$

Equation 23

Finally according to Pritchard and Leylegian, 2011, the Navier_Stokes equations are second to the Bernoulli equation in terms of solving flow problems in fluid mechanics. They form a set of coupled nonlinear partial differential equations when combined with the continuity equation, for u, v, w, and p. to describe many common flows under the assumptions of Newtonian (with a constant viscosity) and incompressible fluid.

3. GENERAL ANALYSIS

This section briefly presents some key flow concepts in fluid mechanics such as inviscid irrotational flows, irrotational two-dimensional Flows, viscous flow in ducts, flow past immersed body, and compressible flow. The section also presents Viscous Flow in Ducts, flow past immersed body, and compressible flow.

3.1 Viscous flow in Ducts

Viscous flow in ducts is a significant phenomenon in fluid mechanics as it is what drives internal flow in pipes and other ducts in chemical and other flow processes. Viscous flow is identifiable in three different flow regimes as shown in figure 18, and these are laminar flow at low Reynolds number, with small natural disburtances, the transition regime with intermediate Reynolds number characterized by intermittent bursts of turbulence, and finally the region of turbulence with high Reynolds number.

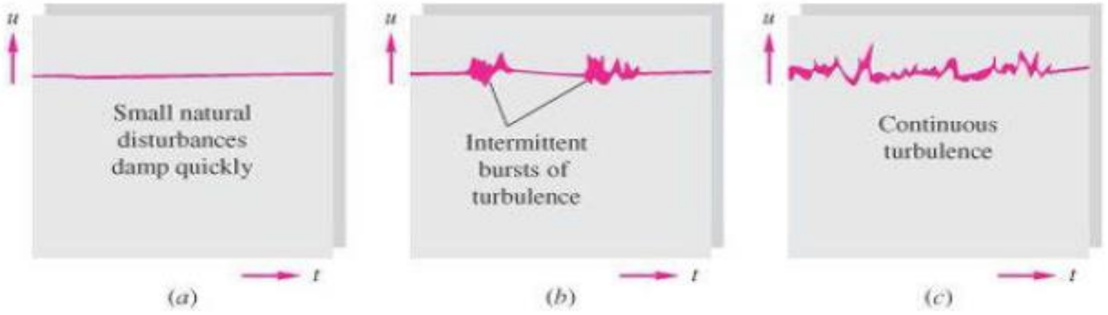


Figure 18: The three regimes of viscous flow: (*a*) laminar flow at low Re; (*b*) transition at intermediate Re; (*c*) turbulent flow at high Re.

The type of flow regimes within a pipe or duct is dependent on the wall roughness or fluctuations in the inlet stream, and this is expressed as a dimensionless parameter called the Reynolds number (Re). The Reynolds number is the ratio of inertial forces to viscous forces, used to categorize fluid flow regimes under significant viscous effect on the velocities or the flow pattern of the fluid. (Rehm and Schubert, 2008)

Mathematically, the Reynolds number, N_{Re}, is defined as

$NRe=\rho\upsilon d\mu$ Equation 24

Where, ρ=density, v=velocity, d=diameter, μ=viscosity

Based on the API 13D, laminar flow falls within Reynolds number less than or equal to 2100 while turbulent flow falls within Reynolds number greater than 2100. (Rehm and Schubert, 2008)

Generally, the following approximate ranges of Reynolds number describe various flow regimes: (Cengel and Cimbala, 2018).

$0 < Re < 1$: highly viscous laminar "creeping" motion

$1 < Re < 100$: laminar, strong Reynolds number dependence

$100 < Re < 103$: laminar, boundary layer theory useful

$103 < Re < 104$: transition to turbulence

$104 < Re < 106$: turbulent, moderate Reynolds number dependence

$106 < Re < \infty$: turbulent, slight Reynolds number dependence

An internal flow in pipes or ducts is generally constrained by the bounding walls, while the viscous effects grow and permeates the entire flow, as shown in figure 19. At the entrance of the duct, the flow approximates an inviscid flow that converges into the tube, resulting into a viscous boundary layer downstream of the inviscid region, that retards the axial flow u (r,x) at the wall while accelerating the centre core flow to maintain the incompressible continuity requirement.

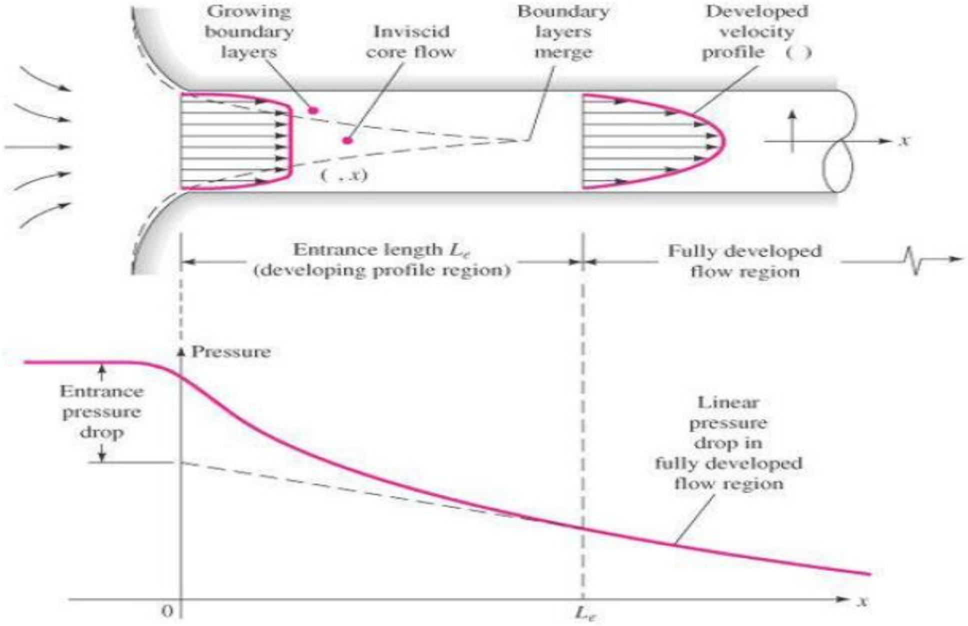


Figure 19. Developing velocity profiles and pressure changes in the entrance of a duct flow. Source: (Cengel and Cimbala, 2018).

The flow becomes an entirely fully developed viscous because the boundary layers merge at this point while the inviscid region gradually disappears at a finite distance from the entrance. One parameter that is critical in in viscous flow in pipes is the head loss or the pipe friction factor, f, which for a fully developed laminar flow, is equals to the change in the sum of pressure and gravity head, that is, the change in height of the hydraulic grade line (HGL), and this is expressed as

$h_f = (z_1\text{-}z_2) + (p_1/\rho g - p_2/\rho g) = \Delta z + \Delta p/\rho g$ Equation 25

This equation is transformed to express a relationship between the head loss and the wall shear Stress, as in equation 26, with the assumption of momentum relation to the control volume, and accounting for applied *x*-directed forces due to pressure, gravity, and shear:

$\Delta z + \Delta p/\rho g = hf = (2\tau_w/\rho g)\ (L/R) = (4\tau_w/\rho g)(L/d)$ Equation 26

It should be noted that the head loss or pipe friction factor, for a fully developed laminar flow, decreases inversely with Reynolds number. Also, in a duct flow there is no work done by the shear stresses at the wall because of zero velocity at the wall, as opposed to work done by pressure forces within the pipe that drives the fluid through the duct. Consequently, the energy generated internally is counterbalanced by viscous dissipation in the interior of the flow. Another factor that affects friction resistance, besides the pipe friction factor, is the surface roughness, though its effect is negligible for laminar pipe flow, it is more pronounced under turbulent flow. (Cengel and Cimbala, 2018).

Viscous flow also occurs in non-circular ducts, such as in flow through parallel plates, flow through concentric annulus, and other non-concentric ducts such as rectangular and isosceles-triangular sections.

The mathematical formulation is therefore different, which relies on hydraulic radius in the computation of its momentum equation:

$\Delta h_f = \Delta p/\rho g + \Delta z = (\tau_w/\rho g) x\ \Delta L/(A/p)$ Equation 27

Where the hydraulic radius is defined as

$R_h = A/p$ = cross sectional area/wetted perimeter Equation 28

This translates to an hydraulic diameter, expressed as

$D_h = 4A/p = 4$ x area/wetted perimeter $= 4R_h$ Equation 29

The wetted perimeter includes all surfaces acted upon by the shear stress. A typical example of non-circular flow is flow through two parrel plates that are a distance 2*h* apart, and the width *b* ≫ *h*, so the flow is essentially two-dimensional. The hydraulic diameter under such condition is expressed as:

$D_h = 4A/p = \lim(b\text{-}\infty)\ 4\ (2bh)/(2b+4h) = 4h$ Equation 30

Which is twice the distance between the plates. Using this, the head loss for fully laminar flow can be approximated as:

$f_{lam} = h_f/(L/D_h)(V^2/2g) = 96\mu/\rho V(4h) = 96/(Re_{Dh})$ Equation 31

Or simply as

$f \approx 64/Re_{D_h}$, though, this simplification would result in about 33 percent lower value.

The same model can also be used to approximate turbulent flow as by introducing the hydraulic diameter Dh = 4h"

$1/(f^{1/2}) \approx 2.0 \log (Re_{Dh} f^{1/2}) - 1.19$ Equation 32

This approximation with hydraulic diameter is quite successful as it closely mirrors the smooth-wall pipe friction law, and therefore can be extended to include other noncircular turbulent flows.

Another case of viscous flow is the flow through concentric annulus as shown in figure 20

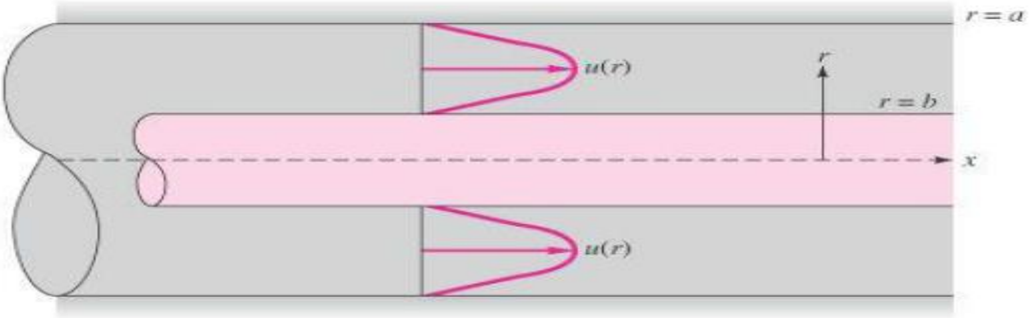


Figure 20: Viscous flow through concentric annulus. Source: (Cengel and Cimbala, 2018).

which a laminar flow in the annular space between two concentric cylinders, with no slip at the inner (*r* = *b*) and outer radius (*r* = *a*), resulting into a parabola velocity profile *u*(*r*) wrapped around in a circle to form a split doughnut. Since this a concentric flow with an inner and a greater outer shear stress, the friction factor will be estimated with respect to the head loss and not on the wall shear:

$f = h_f (D_h/L) (2g/V^2)$ where $V = Q/(\pi(a^2-b^2)$ Equation 33

The hydraulic diameter for an annulus is estimated as:

$D_h = 4\pi (a^2-b^2)/(2\pi(a+b)= 2 (a-b)$ Equation 34

3.2 Flow past Immersed body

Flow past immersed body, also known as external flow, is different from internal flow in pipes or ducts, as the viscous effects in this case are confined within the boundary layers and wakes surrounded by an outer flow region with small velocity and temperature gradients. Under external flow, the flow over the body and the velocity field is greatly influenced by the particular shape of the body. (Cengel and Cimbala, 2018). One of the characteristics of external flow is that they are unconfined and are free to expand regardless of the thickness of the viscous layers Immersed flows over solid bodies are encountered in aerodynamics, hydrodynamics, transportation, wind engineering and ocean engineering. An immersed body, regardless of its shape in a fluid stream, experiences both forces and moments from the flow, that act in all three coordinate axes as shown in figure 21.(White, 2011). The pressure force acts normal to the surface while the shear forces or moment acts parallel to the surface of the body. However, fluid mechanics is not as interested in the distributions of these forces along the entire surface of the body, but more on the resultant effects these forces - the pressure and shear forces have on anybody. Consequently, two main forces are known to act on a body, which are the drag and the lift. The drag is the component of the resultant pressure and shear forces that acts in the direction of flow, while the lift describes the component that acts normal to the flow direction. In figure 21, the drag is shown as the force acting along the axis parallel to the free stream and positive downstream, while the moment about that axis is called the rolling moment. The drag is essentially a flow loss and must be overcome if the body is to move against the stream.

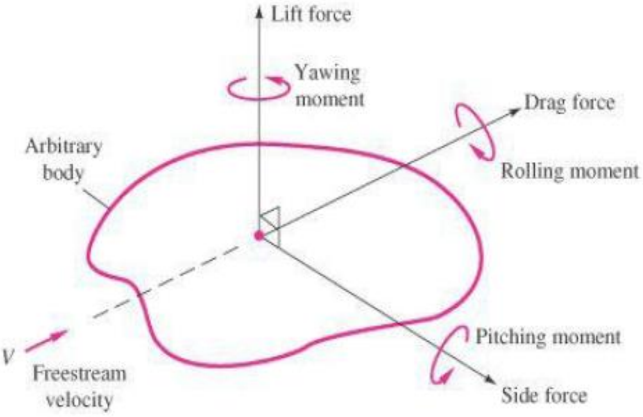


Figure 21. The distribution of the pressure and moment forces in all three coordinate axes.

The Lift is the force that is perpendicular to the drag and usually bears the weight of the body, while the moment about the lift axis is called Yaw. The third and last component is neither a loss nor a gain, and is called the Side Force, while the moment about this axis is called the Pitching moment. The drag force is the main force acting on automobiles, power lines, trees, and underwater pipelines, while the lift force is seen in bird's uplift or at airplane wings. It is also responsible for the upward draft of rain, snow, hail, and the rise of dust particles in high winds; the transportation of red blood cells by blood flow; the entrainment and disbursement of liquid droplets by sprays; the vibration and noise generated by bodies moving in a fluid; and the power generated by wind turbines.

With respect to figure 21, the side force, yaw, and roll all vanish when the solid body has symmetry about the lift–drag axis, such as with airplanes, ships, and cars moving directly into a stream. In such case, the problem reduces to a two-dimensional case: two forces, lift and drag, and one moment, pitch. A further simplification is seen in figure 22 where the body has two planes of symmetry, such as in cylinders, wings, and many other bodies of revolutions.

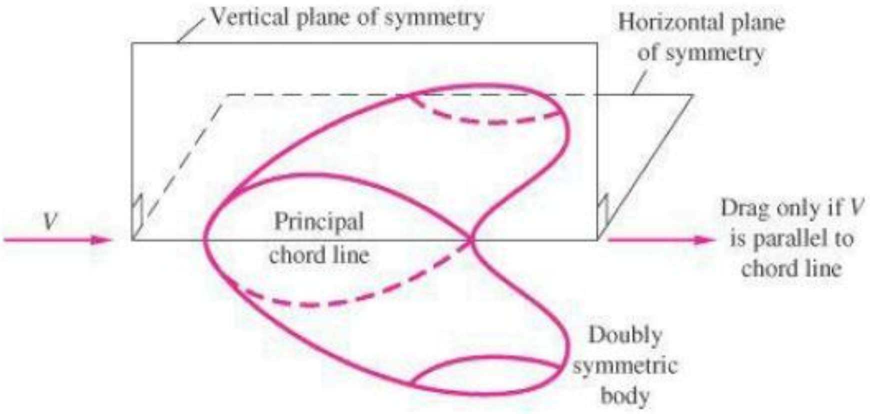


Fig. 22. Dissolution of all forces in a symmetrical body with only drag force and lift forces acting on the body.

The intersection of the vertical and horizontal planes of symmetry forms the principal chord line as shown in figure 22, such that the free stream is parallel to this principal chord line. Under this condition, the body only experiences drag force with no lift, side force, or moments, while the body is observed to exhibit an unsymmetric orientation where all three forces and moments act simultaneously where the free stream does not run parallel to the chord line. (Cengel and Cimbala, 2018). In the case where the body is very long and of a constant cross section, the external flows can be classified as two-dimensional when the flow is normal to the body, and this is seen when wind is blowing over a long pipe perpendicular to its axis. A two-dimensional flow is said to be axisymmetric when the body possesses rotational symmetry about an axis in the flow direction, such as in the case of a bullet piercing through air, with the velocity varying with the axial distance x and the radial distance r. The flow becomes three dimensional if it cannot be modeled as two-dimensional or axisymmetric. Flow across immersed body can also be classified as incompressible flows such as the flows over automobiles, submarines, and buildings and compressible flows as in, flows over high-speed aircraft, rockets, and missiles. Compressibility effects are negligible at low velocities (flows with Ma $\lesssim 0.3$), and such flows can be treated as incompressible with little loss in accuracy. (Cengel and Cimbala, 2018).

Finally, the shape of the immersed body also affects the velocity profile across it, and as such, some immersed body are said to be streamlined, if the shape of the body can be aligned in the direction of the anticipated streamline such as race cars and airplanes, while the body is said to be bluff or blunt such as building when the body tend to block the flow. Technically, streamlined bodies are much easier to be forced through a fluid, the reason why such bodies are of great importance in the design of vehicles and airplanes. (Cengel and Cimbala, 2018).

3.3 Compressible Flow

A compressible flow is when the fluid moves at speed close to its sonic velocity with significant density changes. This is mostly possible with gas flows as liquids would require high pressures in the order of 1000 atm to generate sonic velocities. Gases on the other hand require pressure ratio of only 2:1 to generate sonic flow. will likely cause sonic flow.

Incompressible flow like liquids are characterized by a small Mach number with only small changes in fluid density in all parts of the flow field.

$$Ma = V/a \ll 1$$

where V is the flow velocity and *a* is the speed of sound of the fluid.

The Mach number is the dominant parameter in compressible flow analysis, with different effects depending on its magnitude. whether the flow is subsonic (Ma <1) or supersonic (Ma > 1). The table below shows a distinction between the various ranges of Mach number and the classification of flow regions:

Table 1. Classification of flow regions with Mach Number. Source: White, 2011

Mach Number	Flow Regime
Ma < 0.3	Incompressible flow where density effects are negligible
0.3 < Ma < 0.8	Subsonic flow, where density effects are important, but no shock waves appear
0.8<Ma <1.2	Transonic flow, where shock waves first appear, dividing subsonic and supersonic regions of the flow. Powered flight in the transonic region is difficult because of the mixed character of the flow field
1.2<Ma<3.0	Supersonic flow, where shock waves are present but there are no subsonic regions
3.0 < Ma	Hypersonic flow, where shock waves and other flow changes are especially strong.

3.4 Inviscid Irrotational Flow

Inviscid flow is a flow assumption with zero or negligible viscous-shear and normal stresses. Under this assumption, the only force is the normal stress due to pressure and the normal stress due to viscosity as all the viscous shear-stress terms on the force side of the momentum equations are canceled. Technically, the coefficient of viscosity in an inviscid flow is zero, since the boundary layer on the surface of the body is very thin. The inviscid, incompressible-fluid model technically approximates a perfect fluid since the boundary layer in many practical situations is extremely thin compared to a typical dimension of the body under study. Under this condition, the body shape becomes essentially a geometric shape.

An exception to the inviscid flow, or the conditions where this assumption breaks down occurs when the flow separates with a visible boundary layer that results in a major change in the effective geometry of the body. This is commonly found on wings, and at large angles of attack of a vehicle. However, because wing angle of attack is small for a vehicle at a cruise condition, the separation effects is minimal, and the inviscid model is sustained. However, the inviscid flow model breaks down when the separation is large. Also, since boundary layer is neglected in perfect-fluid theory, the inviscid flow theory does not predict the frictional drag of a body. However, its predictions for low-speed pressure distribution, lift, and pitching moment are valid and useful. (Flandro et al 2012)

Flow is said to be irrotational when the vorticity ω has the magnitude of zero everywhere, resulting in a zero circulation around any arbitrary loop, under the condition that the loop can be spanned by a surface that lies entirely within the fluid. (Fitzpatrick, 2016). However, in a real fluid, viscosity effects will introduce vorticity at a boundary, but at high Reynolds numbers this vorticity will be convected downstream and chiefly confined to the vicinity of the boundary and the wake.). By the definition of irrotational flows, is given as (Graebel, 2007)

$$\boldsymbol{\omega} = \nabla \times \mathbf{v} = 0.$$

Equation 35

However, from Kelvin's circulation theorem, which states that the circulation around any co-moving loop in an inviscid fluid is independent of time, it follows that if an inviscid fluid is initially irrotational then it remains irrotational at all subsequent times. This can be seen from the equation of motion of an inviscid incompressible fluid in the form ((Fitzpatrick, 2016).

$$\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla)\,\mathbf{v} = -\nabla\left(\frac{p}{\rho} + \Psi\right),$$

Equation 36

And because ρ is a constant and given that

$$(\mathbf{v} \cdot \nabla)\,\mathbf{v} = \nabla(v^2/2) - \mathbf{v}\times\ \boldsymbol{\omega}$$

Equation 37

the equation now becomes

$$\frac{\partial \mathbf{v}}{\partial t} = -\nabla\left(\frac{p}{\rho} + \frac{1}{2}v^2 + \Psi\right) + \mathbf{v}\times\ \boldsymbol{\omega}$$

Equation 38

It thus follows from this equation that if the flow pattern in an incompressible inviscid fluid is also irrotational, so that $\omega = \mathbf{0}$ and

$$\mathbf{v} = -\nabla\phi$$

Equation 39

then we can write

$$\nabla\left(\frac{p}{\rho} + \frac{1}{2}v^2 + \Psi - \frac{\partial\phi}{\partial t}\right) = 0,$$

Equation 40

3.5 Computational Fluid Dynamics (CFD)

Engineers readily utilize either experimentation and or calculations when dealing with engineering designs and analysis that involve fluid flow, of these two approaches, the experimentation model usually involves construction of models that are tested in experiments, while the calculation mode only requires solutions to differential equations, which can be done either analytically or using computers (Cengel and Cimbala, 2018). The later solution of the use of computers to solve problems of fluid flow or fluid dynamics is what is referred to as Computational Fluid dynamics or CFD.

Computational fluid dynamics (CFD) involve complex simulations that require a lot of computing time in solving critical flow problems. For instance, simulating a few turns of a propeller can take thousands of computer hours to produce the required model. This is because (CFD) involves the use of a combination of techniques from physics, applied mathematics and computer science, mathematical modeling tools, numerical computation, and software tools to model, predict and visualize fluids flow qualitatively and quantitatively. Evidently, CFD using these tools can easily devise, understand, construct and therefore predict the required scenarios. (Thabet and Thabit (2018).

According to Kundu and Cohen (2008), Computational Fluid Dynamics (CFD) utilizes the conservation laws (conservation of mass, momentum, and energy) in as one of its predictive tool in modeling fluid flow. It also relies on the flow geometry, the physical properties of a fluid, and the boundary and initial conditions of a flow field on one part, and on some sets flow variables such as velocity, pressure, or temperature at selected locations in the domain and for selected times. The prediction also seeks to evaluate the overall flow behavior such as flow rate or the hydrodynamic force acting on an object in the flow. Cengel and Cimbala (2018) also added that CFD helps the engineer to obtain details about the flow field, such as shear stresses, velocity and pressure profiles, and flow streamlines, and in the end, these models are tested and validated with experimental data. Also, CFD has been found to have the capability to handle laminar flow problems without difficulties, while solutions involving turbulent flow are still difficult to handle in its current state, without invoking turbulence models. Cengel and Cimbala (2018) stated that no turbulence model is universal, so that a turbulent CFD solution is only as good as the appropriateness of the turbulence model. Notwithstanding, the standard turbulence models yield reasonable results for many practical engineering problems.

Thabet and Thabit (2018), in their contribution, reported some advantages of using CFD in fluid mechanical solutions and some of these are (i) provides quick results compared to physical modeling techniques that take more time, space and cost to deploy; (ii) allows for in depth queries of any subject matter and helps to create new theoretical developments in several fields for advance developments; (iii) a cost effective measure as it converts actual fluids into digital imagery for better, elongated and uncorrupted analysis. (iv) used for controlled simulations that are not created in real life situations, such as in a nuclear blast or a massive volcano explosion, and finally (v) gives the possibility to analyze different problems which are difficult and dangerous in experimental way

Despite these advantages, Kundu and Cohen (2008), reported some limitations with CFD due to its relies on numerical methods, (finite difference, finite element, finite volume, and spectral methods), that are prone to errors in their approximations. Consequently, CFD predictions are never completely exact, as they are always fraught with errors. Some of these errors include: (i) Discretization error – which is an intrinsic error in all numerical methods, and occurs due to the use of approximations of natural phenomenon with a discrete and finite number of locations in space (grids) or instants of time. This error also depends on the different numerical schemes employed in the solution, and on the distribution of the grids system used in the simulation, even though the same method is employed in the solution. This makes it imperative for engineers to carefully chose a numerical method and a grid to minimize discretization errors. (ii) Input data error occurs since both flow geometry and fluid properties are also only approximations. (iii) Initial and boundary condition error, which is due to crude representation of the initial and boundary conditions of a flow field in the model. For example, flow properties generally are not known exactly and are thus only approximated. (iii) Modeling error occurs as scientific models used in the solution may not accurately describe the actual physical conditions. Examples include turbulence modeling, atmospheric modeling, and problems in multiphase flows, etc.

4. APPLICATIONS OF FLUID MECHANICS

This section presents some of the practical applications of fluid mechanics in engineering and medical sciences.

Fluid mechanics is seen in almost all aspects of human life from turning the water tap in the kitchen that releases water from a network of pipes and valves, to driving cars that rest on pneumatic tires, with hydraulic shock absorbers, and pumping of gasoline through a complex piping system. It is also seen in the flow and distribution of blood through the heart, and oxygen through the lungs and the respiratory system. It also provides answers to several environmental, geotechnical, and structural engineering problems. Fluid mechanics is also involved in air pollution control, water and wastewater treatment, groundwater management and control, and the construction of dams and bridges. (College of Engineering (2021).

4.1 Wind Tunnel

One practical application is the Wind tunnel, a chamber built to examine the characteristics of air flow in contact with solid objects such as aircraft and automobile. The wind tunnel represents a safe and judicious use of the property of fluid mechanics to test the interaction of airflow and solids in relative motion (Mishra et al 2015). It tests when either the aircraft is moving against the airflow, as it does in flight, or when the airflow may be moving against a stationary aircraft. (Science Clarified, 2021).

4.2 Syphon

Fluid mechanics principle is also seen in the design and operation of the Syphon, a long-bent pipe used for carrying water from a reservoir at a higher level to one at a lower level based on the Bernoulli's, principle. (Mishra et al 2015). Under the Bernoulli's Principle, fluid moves from a bigger diameter pipe to a narrower one, at constant volume at a given distance under the same time. The difference in pipe diameter ensures a faster rate of fluid flow through the smaller pipe with greater dynamic pressure to move the same quantity of fluid through the same distance under the same time.

4.3 Airfoil and Airdraft

Bernoulli's principle is also applicable in airfoil, designed for airplane's wing. Airfoils are designed to improve the flying qualities of aircraft and reduce the loads on wind turbines and fan blades. They are also designed to produce maximum lifts that are essentially unaffected by roughness. (Airfoils incorporated, 2000). Bernoulli's principle is also responsible for the creation of a draft in a room, when both a window and the adjoining doors are open at the same time. This situation occurs when inside room temperature is high, whereas the outside temperature is cool, and the only way to cool the room is to open both the window and the door in the hallway to let in the cool breeze into the room. Opening only the window will only cause little temperature change. When the door closes, the room the air pressure in the room increases relative to the outside air pressure, and because as a fluid, air will flow into the room, until the room pressure attains certain point, to stop further inflow of air into the room. So, since the tendency of fluids is to move from a place of high-pressure to low-pressure areas, once the door opens, the relatively high-pressure air in the room will flow automatically into the relatively low-pressure area of the hallway, causing a reduction in the room air pressure, which will now allow in flow of air into the room. With time, the wind from outside will begin to blow in the room. (Science Clarified, 2021)

4.4 Pumps

Pumps which are part of turbomachinery, also operate based on Bernoulli's and Pascal principles. They are devices designed to move fluids under pressure difference, just as the simple syphon hose used to draw gas from car fuel tanks. The piston pump, which is a slightly more complex system, consists of vertical cylinders that enable the piston to travel in an up and down movement. This movement activates a bottom inlet valve that allows fluid to flow into the cylinder and an outlet valve through which fluid flows out. The fluid pump also helps to push the piston up & down. (Mishra et al 2015

4.5 Application in Medical Fields

Fluid mechanics have also helped in medical field to model the flow of blood flow using such assistance devices as Left Ventricular Assist Devices (LVADs), etc. Using these devices, fluid mechanics have helped in modelling of blood rheology and clot formation/lysis in the last 10 to 15 years. (Anand and Rajagopal, 2017). Computational fluid dynamics (CFD), another aspect of fluid mechanics has been used as complementary tool alongside the visualization capabilities of cardiovascular magnetic resonance (CMR) and computed tomography (CT) imaging for decision-making in medical science, especially as it affects the study of the functionality of the heart to detect complex aortic diseases. (Ferreira et al, 2017). Finally, fluid mechanical mathematical models have helped in simulating blood viscosity, a crucial element for any computation of flow fields in the vasculature or blood-wetted devices. The mathematical models for viscosity were built on the foundation of the Krieger model of suspensions, that relies on the dependencies on shear rate, hematocrit, and plasma protein concentrations. The model provided a reasonable compromise in complexity to provide flexibility to account for several factors that affect viscosity in practical applications, while assuring accuracy and stability. (Kameneva and Antaki, 2017)

5. FURTHER APPLICATIONS OF FLUID MECHANICS

Human and plant lives on earth are enveloped in one form of fluid or the other from the air in the atmosphere, to the water in the sea and river and the plants in the soil permeated with moisture. This makes the subject of fluid mechanics critical input into the existence of life on earth. Though fluid mechanics deals with both static and dynamic fluids, the critical component is fluid dynamics that deals with fluid in motion. For instance, the human lungs depends on the flow of air to supply oxygen to the blood stream, also pollutions from industrial systems, are managed and diluted by the continuous motions of air and water through the process of transport phenomenon. This is where the branch of fluid mechanics, called Environmental Fluid Mechanics (EFM) becomes important in balancing the ecosystem.

5.1 Environmental Fluid Mechanics (EFM)

EFM is the scientific study of naturally occurring flows of air and water within planet Earth, especially those that impact on both air and water qualities. Due its importance, EFM also include other disciplines such as meteorology, climatology, hydrology, hydraulics, climnology and oceanography, all of which are concerned with external natural flow within the environment as it does not deal with internal fluid flows in organisms, such as air flow in lungs and blood flow in the vascular system, even though these are also natural flows. EFM also differs from the classical fluid mechanics as the later deals mainly with engineered or artificial fluid flow in chemical and industrial systems.

5.1.1 Geophysical fluid dynamics

Another branch of Environmental Fluid Mechanics is Geophysical fluid dynamics, which is concerned with the physics of atmospheric and oceanic motions on the planetary scale, and due to the strong effects of planetary rotation, the impact of turbulence on geophysical fluid dynamics is relegated to the background. EFM rather focuses on stratification and rotation. Also, while traditional fluid mechanics focuses on hydraulics design and operation, EFM deals more in predictions of environmental-quality parameters that depend on natural fluid flows, as in bedload transport and pollution levels. It also provides decision parameters into how to manage human activities to minimize their impact on the environment. (Gualtieri et al 2012).

5.2 Biofluids Mechanics

Fluid mechanics also find relevance in living organisms as biofluids mechanics, that concern itself with fluid dynamics in biological systems, such as in humans and in plants. Fluid mechanics is responsible for the constant flow and movement of air, water, minerals and nutrients through the human body to maintain organs functionality. The transportation of these organic materials through the human system is done through membranes, cells, tissues, and organs in the body. Also, the body receives these materials and minerals, it also discharges waste or by products through the excretory organs to ensure stability of the body. Maintenance and stability of the human system is strongly dependent on bio transport, which includes the maintenance of pH and body temperature, and the transportation of substances necessary to maintain a stable immunity of the body. Fluid transport is also required in the cochlea in the ear, to enable hearing and motion sensing. All of these bio transport process actually occur at different scales from macro, micro, nano, pico and so on, of which the transportation through cells occurs at micro levels, tissues in micro–macro levels, and organs in macro levels. (Kundu and Cohen, 2008)

Plants also have their share of fluid mechanics in the transportation of nutrients through their cells and tissues, and these can occur either through one of three tissue types, the dermal, ground and vascular tissues. Dermal tissue are those on the outer surface composed of closely packed epidermal cells that secrete waxy material to aid in the prevention of water loss. The ground tissue makes up the bulk of the plant body, and consists mostly of parenchyma, collenchyma, and sclerenchyma cells. Finally, vascular tissues are those that transport food, water, hormones and minerals within the plant. Transport in plants is of three levels: (1). Individual cell’s uptake and loss of water and solutes, (2). short-distance transport of substances from cell to cell at the level of tissues or organs, and, (3). long-distance transport of sap within xylem and phloem throughout the whole plant. Finally, transporation in plants occurs due to fluid gradients in chemical concentration (Fickian diffusion), hydrostatic pressure, and gravitational potential. These three driving potentials are grouped under one single quantity, the water potential. (Kundu and Cohen, 2008)

5.3 Fluid Mechanics and COVID-19 Pandemic

Finally, theory of fluid mechanics has been used to explain how the movement of fluids can help in coming to terms with the present Pandemic caused by the COVID-19 virus. The virus is transmitted by micro-droplets through cough or sneeze, whose motion is governed by the principles of fluid mechanics. According to Pringle, 2020, understanding the way the virus is transmitted can minimize the risk of infection of the virus. For instance, it has been observed that a cough or sneeze transports viral pathogens much further through its multiphase chaotic gas cloud. This has no doubt influenced the thoughts of the World Health Organization (WHO) to acknowledge the possibility of spreading COVID-19 through tiny particles suspended in the air. It has been shown by recent developments in fluid mechanics and epidemiology that turbulent puffs, emitted by sneezing or coughing, has capacity to transport pathogens much further than expected. The characteristics of the “puff” or “plume” that is emitted during breathing, or during cough, or sneeze can provide some insight into how fluid droplets are transported. Fluid droplets inside the “puff” can be distorted when they come in contact with the complex air flow patterns associated with the “puff” or “plume”, which will eventually break up the droplet into several pieces. These small pieces will fall off out of suspension, on many surface, thereby contaminating these surfaces. The break-out smaller particles can also be transported further than the larger ones. According to Pringle, 2020, how far these droplects can travel is also dependent on temperature and humidity , which can actually challenge the WHO's 1-2 metre physical distancing guidelines. This is because, research has shown that these droplets can travel as far as 7 metres, and this does not even consider the micro-droplets that can be transported even further because of building ventilation systems. In an open letter to the World Health Organisation, more than 200 scientists recently accused the organization of underestimating the possibility of airborne transmission of Covid-19.

6. GENERAL RECOMMENDATIONS

This paper has shown how critical the subject of Fluit Mechanics is to all forms of life, and in the growth and development of the industrial economy. Given this vast application, the paper recommends continuous investment in research and development in the study of the various disciplines of fluid mechanics, especially biofluid, environmental and geophysical fluid mechanics. More research are also needed in computational fluid dynamics that can provide concise solutions to the problems of turbulence, since most fluids in life and at industrial scales occur under turbulence regimes.

7. CONCLUSION

While fluid mechanics studies both static and dynamic fluids, as liquids and gases, the aspect of significant relevance is fluid dynamics, since, as we have shown in this paper, the critical roles fluid dynamics plays in both human and plant lives, and in industrial and chemical processes. Fluid dynamics is also responsible for most of the transport and mixing (of materials or properties) that take place in the environment, in vehicles, and in living organisms. In the environment, pollutants such as thermal, particulate and chemicals are transported through fluid dynamics, while as we have shown in this paper, it is also responsible for the transportation of oxygen and blood in human system, and nutrients and carbon dioxide and heat from source to where they are utilized or rejected within the ecosystem. Research in fluid mechanics has as its goal improvement in the ability to predict and control all of these situations, so as to improve the capacity to design devices (for example, aircraft gas turbines, automobile engines) and to regulate (for example, industrial emissions). (Lumley et al 1996).

In conclusion, in view of its wide application, especially during this period of climatic problems caused by emissions of greenhouse gas, and the energy transition, and ofcourse during the COVID-19 Pandemic, more attention need to be given to the subject of fluid mechanics, and investments in research and development to continuously proffer solutions to the problems of fluid dynamics in the entire ecosystem.

REFERENCES

[1] Aerospace, Mechanical & Mechatronic Engg (2005): Aerodynamics for students, http://www-mdp.eng.cam.ac.uk/web/library/enginfo/aerothermal_dvd_only/aero/fprops/cvanalysis/node8.html

[2] Airfoils Incorporated, (2000): Why use new airfoils. www.airfoils.com

[3] Anand, M, and Rajagopal, K.R (2017): A Short Review of Advances in the Modelling of Blood Rheology and Clot Formation, Fluids 2017

[4] Bright Hub Engineering (2009). Classification of Fluid Flow – Uniform Flow and Non-Uniform Flow, 2009, https://www.brighthubengineering.com/hydraulics-civil-engineering/47261-classification-of-fluid-flow/

[5] Cengel, Y.A and Cimbala, J.M (2010): *Fluid Mechanics Fundamentals and Applications,Chapter 4*, *Fluid Kinematics,* Second Edition, McGraw-Hill Education, 2 Penn Plaza, New York, NY 10121, https://slideplayer.com/slide/8366172/

[6] Cengel, Y.A and Cimbala, J.M (2018): *Fluid Mechanics, Fundamentals and Applications* Fourth Edition, McGraw-Hill Education, 2 penn plaza, new york, ny 10121

[7] White, F.M (2011): Fluid Mechanics, Seventh Edition, McGrawHill Company, 2011

[8] College of Engineering (2021) Fluid Mechanics,Southern Illinois University,. https://engineering.siu.edu/civil/facilities/teaching-labs/fluid-mechanics.php#:

[9] Efluids, 2021, Bicycle Aerodynamics, Chapter 7, Streamlines and Streamtubeshttp://www.efluids.com/efluids/pages/bicycle.htm

[10] Ferreira, J.M., Tomas, A.C., Fragata, J and Ramos, H.M (2017): Aorta Ascending Aneurysm Analysis Using CFD Models towards Possible Anomalies, Fluids, 2017

[11] Fitzpatrick, R(2016): Irrotational Flow, https://farside.ph.utexas.edu/teaching/336L/Fluid/node64.html

[12] Flandro, G.A., McMahon H.M., and Roach, R.L (2012): *Basic Aerodynamics, Chapter 4: Fundamentals of Steady, Incompressible, Inviscid Flows, Publisher*: Cambridge University Press

[13] Graebel, W.P (2007): *Advanced Fluid Mechanics*, Academic Press, Elsevier, 2007.

[14] Gualtieri, C., Cushman-Roisin, B and Milhailovic, D (2012): Environmental fluid mechanics: current issues and future outlook. ResearchGate

[15] Hund, S.J., Kameneva, M.V and Antaki, J.F (2017): A Quasi-Mechanistic Mathematical Representation for Blood Viscosity, Fluids,

[16] Janna, W.S (2010): *Introduction to Fluid Mechanics*, Fourth Edition, CRC Press Taylor & Francis Group,

[17] Kundu,P.K and Cohen, I.M (2008): *Fluid Mechanics*, Fourth Edition, Elsevier Inc, Academic Press

[18] Lumley, J.L., Acrivos, A. L., Leal, G, and Leibovich, S (1996*): Research Trends in Fluid Dynamics, book Research Trends in Fluid Dynamics*, the American Institute of Physics, Woodbury, New York.

[19] Mishra V,K., Singh S.K., Patel, S.P., Pandey, S., Dwived, N and Dwived S (2015): Applications of Fluid Mechanics in Different Engineering Fields, *Online Journal of BioSciences and Informatics, Vol: 1, Issue 1, 2015*. https://www.academia.edu/12119005/Applications_of_Fluid_Mechanics_in_Different_Engineering_Fields

[20] Ngo, C.C and Gramoll, K (): Fluid Mechanics – Theory, https://ecourses.ou.edu/cgi-bin/ebook.cgi?doc=&topic=fl&chap_sec=03.2&page=theory

[21] Pringle, J (2020): How does coronavirus spread? The principles of fluid mechanics hold the answer, https://scroll.in/article/967097/how-does-coronavirus-spread-the-principles-of-fluid-mechanics-hold-the-answer

[22] Pritchard, P. J and Leylegian J.C (2011): *Introduction to Fluid Mechanics*, Eight Edition. John Wiley & Sons, Inc.,

[23] Ray Solutions (2018): Fluid Mechanics, https://www.rgpvonline.com/answer/fluid-mechanics/7.html,

[24] Reynolds' Number, https://www.sciencedirect.com/topics/engineering/reynolds-number, Bill Rehm, ... Jerome Schubert, in Managed Pressure Drilling, 2008

[25] Science Clarified (2021): Fluid Mechanics - Real-life applications Read more: http://www.scienceclarified.com/everyday/Real-Life-Chemistry-Vol-3-Physics-Vol-1/Fluid-Mechanics-Real-life-applications.html#ixzz6nruFZi00

[26] Srivastava, A (2020): Fluid, Fluid Everywhere: The Fascinating Tale of Fluid Mechanicshttps://medium.com/@apeksha.0503/fluid-fluid-everywhere-the-fascinating-tale-of-fluid-mechanics-aead66847033,

[27] Thabet, S and Thabit, T.H (2018): Computational Fluid Dynamics: Science of the Future *International Journal of Research and Engineering ISSN: 2348-7860 (O) | 2348-7852 (P) | Vol. 5 No. 6 | June 2018 | PP. 430-433,*