CONCEPT OF FLUID MECHANICS

Dr. Avdesh Singh Pundir
Ramachandran T

BOOKS
ARCADE

Concept of Fluid Mechanics

Concept of Fluid Mechanics

Dr. Avdesh Singh Pundir
Ramachandran T

BOOKS ARCADE
KRISHNA NAGAR, DELHI

Concept of Fluid Mechanics

Dr. Avdesh Singh Pundir
Ramachandran T

BOOKS ARCADE
Regd. Office:
F-10/24, East Krishna Nagar, Near Vijay Chowk, Delhi-110051
Ph. No: +91-11-79669196, +91-9899073222
E-mail: info@booksarcade.co.in, booksarcade.pub@gmail.com
Website: www.booksarcade.co.in

Year of Publication 2023

International Standard Book Number-13: 978-81-962948-1-6

CONTENTS

CHAPTER 1

INTRODUCTION OF FLUID MECHANICS

Dr. Avdesh Singh Pundir,
Associate Professor, Department of Chemical Engineering,
School of Engineering & Technology, Jaipur National University, Jaipur, India,
Email Id- dr.avdeshpundir@jnujaipur.ac.in

The study of all fluids in both static and dynamic conditions is the focus of fluid mechanics. The interaction between forces, movements, and statical states in a continuous material is the subject of fluid mechanics, a subfield of continuous mechanics. Surface energy, fluid statics, flow in enclosed bodies or flow around bodies (solid or not), flow stability, and many other issues are dealt with in this subject field. In reality, a fluid mechanics issue arises in practically every movement a human takes. Additionally, rather than being a clear division, the line between solid mechanics versus fluid mechanics is more of a fuzzy area. When examined closely, glass, which initially seems to be a solid, is actually a liquid with a high viscosity. The changing glass height in high windows in European churches after a century is evidence of the glass's "liquidity." The glass's bottom portion is thicker than its top. Grains and materials like sand (also known as quick sand) should be handled like liquids. These substances have the potential to drown humans, as is well documented. Below the mushy zone, materials like aluminium also act like a liquid, much like butter. Additionally, mixing material particles that "behave" as solids with liquids produces a combination. The majority of the material in this book is restricted to basic and (mainly) Newtonian (rarely power fluids) fluids that will be specified later once it was proven that the limits of fluid mechanics are not crisp.

The study of fluid mechanics spans a wide range of disciplines without any discernible distinctions. Laminar flow and turbulent flow are two types of flow that researchers differentiate from one other. It is also possible to differentiate between single-phase flow and multi-phase flow in terms of fluid mechanics (flow made more than one phase or single distinguishable material). Because fluid may undergo a phase transition (condensation or evaporation) in the middle or during the flow and transform from a single phase flow to a multi-phase flow, the final boundary (like that of all the boundaries in fluid mechanics) is not crisp. Additionally, a flow that contains two phases (or components) might be thought of as a small generator (for example, air with dust particle).

The research must draw artificial boundaries across domains since it has become obvious that the limits of fluid mechanics are not well defined. Then, using dimensional analysis, it will be possible to explain why, in some situations, one distinct region or principle is more important than another and some impacts may be disregarded. Or maybe a broader model is required since the issue is affected by more variables. According to the author's own experience, one of the primary issues is the lack of information and the inability to determine where the situation really was. Engineers at the software business EKK Inc., for instance, examined the flow of an entire still liquid while assuming a sophisticated turbulent flow model. Such ludicrous analyses are typical among engineers who are unsure about the best model to use. This book's explanation of the appropriate model is one of its primary objectives. The streamlined private instances must be discussed before moving on to the limits.

The presentation of a fluid mechanics introduction may be done in one of two ways. The essential governing equations are introduced in the first method, which is then followed by safety, turbulence, and boundary layer. The second method focuses on integral analysis, which is then followed by differential analysis and then empirical analysis.

The necessity for a water supply led to the need for some knowledge of fluid dynamics. For instance, individuals became aware that wells must be developed and rudimentary pumping equipment must be built. Later, a need to address waste (sewage) was generated by a big population, and some fundamental knowledge was developed. People eventually came to the realisation that water can be utilised to power devices and move objects. Aqueducts were built as cities grew to bigger sizes. These aqueducts were at their largest and grandest in the ones in China and the City of Rome.

All ancient knowledge, with the exception of Archimedes' (250 B.C.) understanding of buoyancy principles, may be distilled down to the application of instincts. For instance, bigger tunnels might be constructed for a bigger water supply, etc. Even with the significant demand for transportation and water supplies, there were no estimates. By creating the first chambered canal lock close to Milan, Leonardo Da Vinci (1452-1519) produced the first advancement in fluid mechanics. He also made multiple efforts to research bird flying and established some theories about the forces' beginnings. Following his first work, Galileo, Torricelli, Euler, Newton, the Bernoulli family, and D'Alembert made significant contributions that helped advance understanding of fluid mechanics (hydraulics). Theory and experiments at that time were not entirely consistent. In his statement that "The theory of fluids must unavoidably be built upon experiment," D'Alembert accepted this truth. For instance, the notion of the perfect liquid, which causes motion without resistance, contradicts with the truth.

The "D'Alembert paradox" refers to this difference between theory and practise and helps to highlight the limits of theory as a stand-alone approach to resolving fluid issues. Similar to thermodynamics, two schools of thought were developed: one held that the answer would come from the theoretical side alone, and the other held that the solution was the top - down method (experimental) element of fluid mechanics. Theoretical contributions from Euler, La Grange, Helmholtz, Kirchhoff, Rayleigh, Rankine, and Kelvin were significant. The "experimental" group included Brahms, Bossut, Chezy, Dubuat, Fabre, Coulomb, Dupuit, d'Aubisson, Hagen, and Poisseuille, with a focus on pipes and open channels. Real fluid motion's governing equations were originally developed by Navier at the molecular level and then by Stokes from a continuous point of view in the middle of the nineteenth century resulting in a convergence of the experimental and theoretical schools of thinking. But humans cannot give up control, just as in thermodynamics. As a consequence, "strange" names like hydrodynamics, hydraulic systems, gas dynamics, and aeronautics were born.

Because of their tremendous complexity, the Navier-Stokes equations, which describe flow, or even the Euler equations, were thought to be unsolvable in the middle of the nineteenth century. There were two results from this issue. Theorists attempted to reduce the equations and reach approximations of solutions that represented certain instances. Examples of such work include the Kutta-Joukowski circulation theory of lift, Lanchester's idea of circulatory flow, and Hermann von Helmholtz's concept of vortexes (1858). The experimentalists simultaneously put up a number of correlations for various fluid mechanics issues, such as viscosity by Darcy, Weisbach, Fanning, Ganguillet, and Manning. The obvious occurred without theoretical direction; empirical formulae were produced by fitting curves to experimental data (or sometimes, just by reporting the findings in tabular form), and these formulas made very little sense in terms of the link between physics and qualities.

The growth of several businesses at the end of the 20th century led to a desire for solid scientific knowledge that could be used to a variety of liquids rather than a formula for each liquid. This requirement, together with a number of fresh ideas, such as Reynolds' theoretical and experimental studies, Rayleigh's creation of dimensional analysis, and Froude's notion of using models, transform the field of fluid mechanics. The boundary layer theory of Prandtl, which combines modelling and dimensional analysis to produce contemporary fluid mechanics, is perhaps the most revolutionary notion that has an impact on fluid mechanics. As a result, Prandtl is widely regarded as the founder of modern fluid mechanics. This idea provides the mathematical foundation for several approximations. Thus, fluid mechanics was developed into today's contemporary science by Prandtl and his pupils Blasius, von Karman, Meyer, and Blasius, as well as a number of other people including Nikuradse, Rose, Taylor, Bhuckingham, Stanton, and many more.

Although the foundations were still well understood, how they were computed altered after World War Two. The discipline has altered as a result of the arrival of personal computers in the 1960s and their increased power. Numerous open source applications are available that can assess a wide range of fluid mechanics scenarios. The use of numerical tools now allows for the analysis of numerous issues and the production of acceptable solutions. In many instances, these tools are able to capture all the necessary characteristics and offer a sufficient description of the physics. However, there are several additional situations in which a meaningful conclusion cannot be drawn from numerical analysis (trends). For instance, no weather forecast algorithm can provide findings of acceptable engineering quality (where the snow will fall with an accuracy of 50 kilometres). Building an automobile with this level of precision is disastrous). The best case scenario is that these programmes are only as good as the input. As a consequence, presuming turbulent flow for still flow only leads to inaccurate findings.

History of Fluid Mechanics

The availability of water for home use and crop irrigation was one of the earliest technical challenges that humanity encountered when cities were built. Only with plentiful water can we maintain our modern lifestyles, and it is obvious from the investment in archaeology that every prosperous prehistoric society made the development and upkeep of water infrastructure. The aqueducts of Rome, The most well-known examples include ones that are still in use. However, from a technical standpoint, Pergamum in modern-day Turkey may have produced some of the most spectacular engineering.

The Greek mathematician Archimedes produced the earliest acknowledged contribution to the science of fluid mechanics (285–212 BC), in the first nondestructive test in human history, the buoyancy principle was developed and used to ascertain the amount of gold in King Hiero I's crown. The Romans constructed impressive aqueducts and taught many subjugated nations about despite the advantages of clean water, they generally lacked a solid grasp of fluids theory. The middle Ages saw a gradual but steady use of fluid machines. Progressively increased. For dewatering, elegant hydraulic actuators were created. Mines, grain mills, forges, and windmills all of which were perfected metal, as well as for other uses. Significant labor was being accomplished for maybe the first occasion in mankind's history without the assistance of a muscle powered by a person. These inventions are often attributed with making it possible for a person or animal. The industrial revolution that followed. Once more, those responsible for much of the advancement unknown.

Athough several sources have reported the gadgets themselves specialised authors like Georgios Agricola. Machinery, but more significantly, the development of the scientific method and embraced over all of Europe. Galileo Galilei, Simon Stevin (1548–1617), and Edme Mariotte (1620–1684), Evangelista Torricelli, and (1564–1642) One of the first to use the technique on fluids were (1608–1647) as examined vacuums and hydrostatic pressure distributions. That project was the excellent mathematician Blaise Pascal (1623-) combined and improved1662). The continuity concept for fluids was initially articulated by the Italian priest Benedetto Castelli (1577-1644). Besides inventing his equation for objects, Sir Isaac Newton (1643–1727) (1643–1727) extended his principles to fluids and investigated the resistance and inertia of fluids, free jets.

Theory of fluid mechanics has advanced up until the end of the Due to fluid qualities, the seventeenth century had minimal effect on engineering. Parameters were not well specified, and the majority of the hypotheses were abstractions which, for the sake of design, could not be measured. That was about to alter with Riche de Prony's creation of the French engineering school (1755–1839). Prony, who is still renowned for his brake-powered power meter, and his associates at the Ecole Ponts et Chaussees and the Ecole Polytechnique in Paris incorporated mathematics and philosophy of science into engineering for the first time, Curriculum, which was adopted as the standard worldwide.

By the middle of the nineteenth century, significant advancements had begun, numerous fronts. The physician Jean Poiseuille had precisely examined the flow of several fluids in capillary tubes while in Germany. In pipes, Gotthilf Hagen distinguished between flow regimes. The work was carried on in England by Lord Osborn Reynolds (1842–1922) who also created the dimensionalless number which bears his name. Similarly, George Stokes' early work might be compared to that of Naiver.

The general equations of flowing fluid with frictional (finished in 1903) their names, please. William Froude (1810–1879) nearly did everything. Created the processes shown the worth of testing physical models. The development of fluid theory was important in the late nineteenth century. by engineers and scientists from Ireland and England, in addition to Lord Kelvin (1824–1907), William Thomson, Reynolds and Stokes, and William Sir Horace Lamb, Lord Rayleigh (1842–1919), and Strutt (1849–1934). These researchers looked on many different issues, such as dimensional analysis, asymmetric channel flow, vortex movements, cav, and waves. In an authors studies also examined the connections between fluid mechanics, heat transfer, and thermodynamics.

One may argue that the middle of the 20th century was the height of fluidity applications in mechanics. The challenges at hand were suitable for existing ideas. Hand, and the parameters and fluid characteristics were well stated. These enabled the aviation, chemical, manufacturing, and water industries to grow significantly. Resources industries, each of which changed the course of fluid mechanics. The American invention of the microcomputer dominated fluid mechanics activities in the latter half of the 20th century. The capacity for our civilization has benefited from efforts to address difficult challenges, such global climate modelling or to improve the design of a wind turbine. The fluid mechanics pioneers of the eighteenth century could never have imagined.

Vortex dynamics developed as a branch of fluid mechanics during the course of the following century, consistently occupying at least a significant chapter in treatises mostly on topic. As a result, both G. and H. Lamb's well-known Hydrodynamics (6th ed., 1932) book dedicate an entire book to vorticity and vortex dynamics. Introduction to Flow Of fluids by K. Batchelor (1967). Soon, whole treatises were devoted to the motion of vortices. H. The Theory of Tourbillons by Poincaré (1893), H. Lessons on the Theory of Tourbillons by Villat (1930), C. The Kinematics of Turbulent kinetic energy by Truesdell, published in 1954, and P. Vortex Dynamics (1992) by G. Saffman can be suggested. Early on, spins, vortex motion, vortex dynamics, and vortex flows were the subject of separate sessions at scientific conferences. Later, the topic was the focus of whole meetings.

The first to use experimental scientific methods in fluid mechanics, notably in the area of fluid statics, such as for estimating specific weights, were Islamicate scientists, mainly Abu Rayhan Biruni (973–1048), and subsequently Al–Khazini (fl. 1115–1130). They utilised ratio theories and infinitesimal techniques in mathematics, as well as algebraic and fine-grain computation methods, to advance the subject of fluid dynamics.

Kinds of Fluids

Some distinguish between fluid and solid based on how they respond to shear stress. While the solid displays a limited deformation that does not alter over time, the fluid distorted constantly and permanently under shear force. Additionally, it is said that fluids can never regain their former shape after being deformed. Three categories of materials result from this differentiation: solids, liquids, and everything else. This experiment produces a novel class of materials that exhibits dual behaviors; within some bounds, they act as solids and, within other bounds, as fluids Rheology is

Generally speaking, liquids and gases make up the fluid. The primary distinction between the states of liquids and gases is that although liquids have a nearly fixed volume, gases will fill the whole space. Even if this disparity isn't very acute, it may be deemed sharp for the majority of reasons. Above the critical point, the differences between a gas phase and a liquid phase are essentially negligible. However, below the critical limit, a 1000% increase in water pressure only results in a 1% decrease in volume. For instance, a volume change of more over 5% necessitates a pressure change of tens of thousands of percent. Therefore, the pressure won't affect the volume if the change in pressure is substantially smaller than that. Any change in pressure immediately influences the volume in the gaseous phase. Liquid cannot fill the space; only gas can. No free interface or surface exists for gas (since it does fill the entire volume).

In this debate, a number of quantities need to be addressed. In physics, force was discussed as the first [N] is the measurement unit. It is important to keep in mind that force is a vector since it has a direction. The area is the second quantity we'll talk about here. Although this amount was covered in physics class, it is used here to refer to the direction of the region. Perpendicular to the region is the direction. There is no one direction in a three-dimensional object's area. Therefore, these types of issues should be dealt with locally and infinitesimally.

Force per area, a conventional measure, now has a new meaning. Tensor is the term used to describe the outcome of dividing a vector by another vector. The tensor will need to be disassembled at this point since the focus of this book is on physics. The examination of the mathematical significance is offered later (later version). In this case, the pressure has three components: one perpendicular to the area and two parallel to it. Pressure is the term for the pressure portion in the area direction (what a terrific way to confuse people!). Shear stresses are the name given to the other two elements, as shown in Figure 1.1

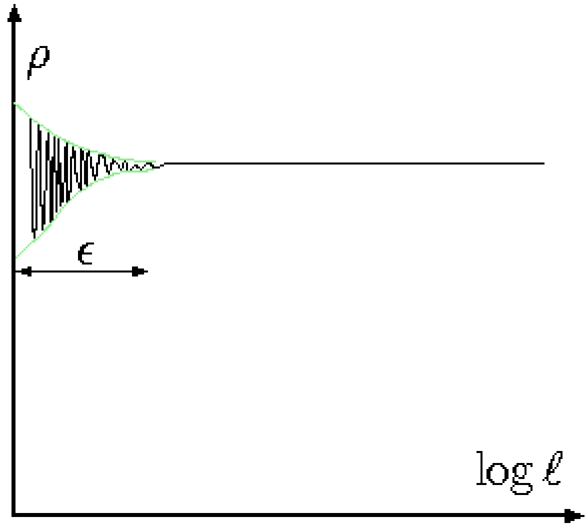

Figure 1.1 represents size of the region.

The study of how gases and liquids behave, particularly the forces they generate, is known as fluid mechanics. Fluid dynamics is a topic that interests many scientific fields. For instance, in order to anticipate the weather, meteorologists attempt to predict the velocity of the planet's fluid atmosphere. In an effort to find a practical way to use the energy produced by nuclear fusion processes, physicists are researching the movement of extremely hot gases across magnetic fields

The study of fluids in motion and at rest is known as fluid mechanics. A substance that constantly deforms below a constant stress is referred to as a fluid. The five relationships kinematic, stress, conserving, regulating, and fundamental are the most helpful in fluid mechanics issues. The analysis of fluid mechanics issues can be varied based on the choice of the system or subsystem and the size of interest, which determine the simplification of quantitative variables. The first branch of physical science that studies both stationary and moving objects caused by external forces. The mechanical specialty that statics is the branch of science that statics is the study of bodies at rest.

Dynamics is the study of moving bodies. The fluid mechanics subcategory is described as the branch of science that examines how fluids behave both in motion and at rest, as well as how they interact with solid objects at the boundary, or other liquids. Fluid dynamics is another name for fluid mechanics a specific instance of movement with zero speed, fluids at rest may be used to understand dynamics.

Even fluid mechanics is broken down into many areas. Examination of the motion of fluids that are virtually highly viscous (such as liquids, notably water, plus gases at slower revs) is commonly referred to as fluid mechanics. Hydraulics, which deals on liquid flows in pipes or open channels, is a subclass of hydrodynamics. Gas dynamics examines the movement of fluids that experience large density fluctuations, such gas flow high-speed thru nozzles. The subject of aerodynamics is the passage of gases, notably air, at high or moderate speeds over objects like rockets, cars, and aeroplanes. Other specialised fields that deal with naturally occurring flows include meteorology, oceanography, and hydrology

The area of fluid mechanics does indeed have a broad application and is crucial to many engineering and scientific disciplines. The emphasis of the current course is on applying fundamental fluid mechanics concepts to address practical situations. The process of obtaining each controlling equation from the underlying concept is given special consideration. There is a nicely balanced presentation of physical ideas, mathematical operations alongside examples and workout problems of practical value. After finishing the course, the learners will possess a solid foundational grasp of fluid mechanics' fundamental concepts and be able to use those principles to analyse fluid mechanical systems. It is a branch of continuum mechanics, a subject which approaches matter without using the information that it is decided to make out the of atoms, that is, it manufacturers matter from a macroscopic viewpoint instead of from a microscopic viewpoint.This study area deals with many and diversified problems, such as:It is an offshoot of continuum mechanics, a subject which models make a difference without using the information that it has been made out of atoms.

1. Skin tension,
2. Fluid dynamics
3. Flow in or around bodies to surround them (solid or otherwise)

An important topic in civil, mechanical, and chemical engineering is fluid mechanics, the field of science that concerned with the diagnosis of flows (liquids and gases) in a state of relaxation or motion. Fluid dynamics, fluid kinematics, and fluid statics are some of its branches. Fluid refers to a material that flows. Fluids are any substances that are liquid or gaseous. Because they are utilized in so many different applications, freshwater, oil, and other materials are crucial to our daily lives. For instance, oil is used to lubricate autos, water is utilized to generate energy in hydroelectric and thermal power plants, and water serves as a refrigerant in nuclear power plants. In some ways, a typical home resembles an exhibit hall with fluid mechanics applications. The pipe systems for natural gas, cold water, and drainage for a single home as well as the entire building are largely.

The density is a characteristic that necessitates continuous liquid. The density may be altered and is dependent on both time and place (location), but it must also possess a continuing quality. It doesn't follow that there can't be a sudden, dramatic shift in density. It made reference to the fact that sample size has no effect on density. As a result, the density is defined as follows:

lim Delta is selected to ensure that the continuous assumption is maintained, i.e., that it has not reached or been reduced to a size at which atoms or molecular statistical computations become meaningful

Shear Stress

The pressure tensor includes the shear stress. However, it will be handled separately in this section and throughout much of the book. The ratio of the force operating on an area in the direction of the force acting perpendicular to the area is referred to as the shear stress in solid mechanics. Fluid cannot draw through a solid surface directly, unlike solids. Think of a liquid that experiences shear force between two plates.

Typically, the top plate velocity will be U = f (A, F, h) labelintro:eq:uPlate endalign. Where A stands for area, F for force, and h for the separation between the plates. It has been shown via solid mechanics research that as force per area rises, so doe's plate velocity. Increasing height will, up to a point, enhance velocity, according to experiments. Think about sliding the plate with either no lubrication ($h sim 0$) (which produces a lot of force) or a lot of lubricant (smaller force). The small distance analysis is relevant in this topic since the goal is to create a differential equation.

Viscosity

Temperature has a significant impact on viscosity. The viscosities of liquids and gases, however, are affected in the reverse way by temperature fluctuation. Their fundamentally distinct mechanisms for producing viscosity properties account for the disparity. In contrast to liquids, which have more compact molecules and stronger cohesion, gases have sparse molecules and weak cohesion. As a consequence, molecular mobility in gases that is normal to the overall direction of flow results in the exchange of momentum across layers, which opposes the flow. The viscosity of gases will rise with temperature since it is known that this molecular activity increases with temperature. The kinetic theory's concerns lead to this line of thinking. According to this idea, the square root of temperature directly affects the viscosities of gases. In liquids, the pressure forces between the molecules outweigh the momentum exchange brought on by molecular mobility. Thus, the strength of these forces of attraction has a significant influence on viscosity. Liquid viscosities drop as temperature rises because these forces quickly diminish as temperature increases.

Non–Newtonian Fluids

A fluid that deviates from Newton's law of viscosity—constant viscosity regardless of stress—is said to be non-Newtonian. When subjected to force, the viscosity of non-Newtonian fluids may vary, becoming either more flexible or more solid. For instance, ketchup is a non-Newtonian fluid because shaking causes it to become runnier. Non-Newtonian fluids include various salt solutions and molten polymers, as well as many everyday things such custard toothpaste, starch solutions, corn starch, paint, blood, 2 cups, and shampoo. Most often, the viscosity of non-Newtonian fluids depends on the shear rate or shear rate history (the progressive deformation caused by shear or tensile stresses). Nevertheless, certain non-Newtonian fluids with shear-independent viscosities continue to display typical stress-difference patterns or other non-Newtonian behaviours. In a Newtonian fluid, the coefficient of viscosity serves as the proportionality constant, and the relationship between the shear stress and shear rate is linear, traveling through the origin. The relationship in between shear increasing shear rate differs in a non-Newtonian fluid. Even a time-dependent viscosity may be seen in the fluid. Therefore, it is impossible to determine a constant coefficient of viscosity.

Although viscosity is often employed in mechanics to explain a fluid's shear characteristics, non-Newtonian fluids cannot always be adequately described by this notion. They may be examined most effectively using a number of additional rheological characteristics that link stress and strain rate tensors under several flow situations, such as oscillatory stress or extensional flow, and are determined by various instruments called rheometers. Tensor-valued constitutive equations, which are often used in continuum mechanics research, are a superior tool for understanding the characteristics.

Non-Newtonian fluid refers to any fluid, as well as its higher powers and derivatives, in which the rate of deformation is not proportional to the stress. It's critical to comprehend the definition of fluid before delving into non-Newtonian fluid in further depth. A material that flows while being affected by shear forces is referred to as a fluid. Fluids include gases, liquids, and plasma, among others. In certain ways, plastic solids are termed fluids. Based on the connection between shear stress and the rate of strain and its derivatives, fluids may be categorised as one of the following:

Shear rate or shear rate history determines the viscosity of the majority of non-Newtonian fluids, which is the progressive deformation brought on by shear or tensile stresses. The shear-independent viscosity of certain non-Newtonian fluids, however, exhibits normal stress variations or other non-Newtonian properties. The fluid's viscosity might possibly alter over time. As a consequence, it is impossible to establish a constant viscosity coefficient. Let's examine non-Newtonian fluid in more detail. NonNewtonian fluids have shear stress that is not proportional to deformation rate. Although we won't go into detail about it in this article, many common fluids display non-Newtonian behaviour. Toothpaste and Lucite 6 paint are two well-known examples. When sheared by brushing, the latter is quite "thick" in the can but becomes "thin" when sheared. Toothpaste acts like a "fluid" when pressed from the tube. When the cap is removed, however, it does not run out on its own. There is a yield stress below which toothpaste acts like a solid. Our notion of a fluid is strictly limited to materials with zero yield stress. Non-Newtonian fluids are generally classed as either time-independent or time-dependent. The rheological shows examples of time-independent behaviour. To describe the observed relationships between ryx and du/dy for time-independent fluids, several empirical equations have been presented. They may be effectively described for many engineering purposes by the bell curve model, which becomes

$$\tau_{yx} = k\left(\frac{du}{dy}\right)^n$$

Solids, Liquids and Gases

Atoms and molecules make up fluids. The distance between molecules varies by orders of magnitude depending on the fluid phase (gas, liquid, or supercritical), being longest in the gas phase and lowest in the liquid phase. The flow cannot be thought of as a continuum when the space between molecules or mean free path of the flowing media approaches the typical size of the flow device. Molecules arrange themselves into a regular grid and oscillate around an equilibrium point in solids. The molecules are strongly attracted to one another in this condition, and the molecules' kinetic energy cannot overcome this pull. The substance melts and then turns into a liquid when the molecules are given enough energy, such as through heating. The additional heat causes the molecules to gather kinetic energy and begin to move in an erratic manner. The mean molecular lengths at these two phases, or the density in liquids and solids, do not significantly vary from one another. The density dramatically decreases when the liquid vaporises and transitions to the gas phase because the molecules may now move freely between intermolecular collisions.

Fluid

A material that continuously deforms (flows) under an applied shear stress is referred to as a fluid, regardless of the size of the applied stress. A solid, however, can withstand an applied force by kinetic deformation. It is generally agreed that fluids include liquids, gases, plasmas, and to some degree, plastic solids. The form of a perfect fluid is determined by the geometry of its container since it has no internal barrier to shape change. In contrast to gases and plasmas, which do not form a free surface but instead expand to fill the whole volume of the container, liquids create a free surface.

There is no disputing the significance of flow phenomena. Flow phenomena might be fully or partly involved in technical applications or natural phenomena. It may be accomplished throughout a wide variety of time frames. Blood flow and atmospheric flow are two instances of this kind. Humanity learnt to use flow phenomena as a tool-making species. Therefore, people who work with flowing matter should have a stronger theoretical foundation and be able to apply methods for experimental and numerical inquiry.

Fluid mechanics has been crucial to human existence. As a result, it also drew in a lot of interested individuals. Systematic theoretical work has been done even in the history of ancient Greece. Beginning in the 16th century, governing equations for fluid flow were being developed. The most basic version of the conservation principles for mass, momentum, and energy was previously known in the 18th and 19th centuries. The twentieth century saw advances in theoretical, experimental, and most recently numerical fields. Most governing equation solutions for specific circumstances were given in the theoretical area. Fluid characteristics and flow velocities may now be measured experimentally. The numerical treatment of fluid mechanical issues was made possible by the advancement of computers, opening up new avenues for investigation. The development of new experimental and numerical tools, as well as their use in the creation of new technologies, are thought to be where activities will be most intense in the 21st century.

Scope of Fluid Mechanics

Any system in which a fluid serves as the working medium must be analysed, and knowledge of and comprehension of the fundamental principles and ideas of fluid mechanics are crucial. have a tonne of examples. The concepts of fluid mechanics must be used in the design of almost all modes of transportation. Surface ships, submarines, cars, and supersonic and subsonic aircraft are all included. Automobile makers have recently given aerodynamic design greater thought. For both racing car and boat designers, this has been the case for a while. On the principles of fluid mechanics, propulsion systems for both toy rockets and space travel are designed. The Tacoma Narrows Bridge collapse in 1940 is proof of the potential repercussions of disregarding the fundamentals of fluid mechanics.

To calculate the airflow on and flow fields around buildings and structures, model studies are often carried out nowadays. These include research on smokestacks, skyscrapers, baseball stadiums, and retail centres. It is obvious that understanding the fundamentals of fluid mechanics is necessary for the design of all varieties of fluid equipment, including pumps, fans, blowers, compressors, and turbines. In fluid mechanics, lubrication is a very significant application. Additional technical issue areas needing an understanding of fluid mechanics include the design of pipeline systems, heating and ventilation systems for individual residences and big office buildings, and more. The body's circulatory system is mostly a fluid system. It is not unexpected that the design of breathing machines, heart-lung machines, artificial hearts, blood replacements, and other similar devices must adhere to the fundamentals of fluid mechanics. Even some of our leisure activities have a direct connection to fluid mechanics.

The fundamentals of fluid mechanics may be used to explain why golf balls slice and hook. There are many ways that fluid mechanics may be used in daily life. Our major argument is that fluid mechanics is not only a topic studied for academic purposes, but also has a significant impact on current technologies and daily life.

It is obvious that we are unable to fully address even a tiny portion of these and other unique fluid mechanics issues. This text's focus is instead on outlining the fundamental physical principles and rules that serve as the foundation for investigating any fluid mechanics issue.

1. The mass is conserved.
2. Newton's second law of motion is number two.
3. The angular momentum principle
4. Thermodynamics' first law,
5. The thermodynamic second law.

Not every fundamental law is necessary to resolve a particular issue. On the other hand, in many issues it is required to include additional relations in the analysis that characterize the behavior of fluids' physical characteristics under specific circumstances.

For instance, you may have studied the characteristics of gases while studying thermodynamics or fundamental physics. For many gases under typical circumstances, the ideal gas equation of state, p = pRT, is a model that connects density to the temperature and pressure. The gas constant, is found in Eq. R values for various common gases are provided in Appendix Ap and T stand for absolute pressure and temperature, respectively; p is density (mass per unit volume). The application of the ideal gas equation of state

Utilised the free-body diagram (system approach) a lot in mechanics classes. This made sense considering that you were dealing with a hard body that was simple to identify. However, in the field of fluid mechanics, we often focus on the movement of fluids via machinery like compressors, jets, pipelines, injectors, and so on. In these situations, it is challenging to concentrate on a constant, measurable amount of mass. To analyse, it is far more practical to concentrate on the area of space where the fluid moves. As a result, we are using the control volume strategy.

An arbitrary space volume through which fluid flows is referred to as a control volume. The control surface is the geometric limit of the control volume. The control surface might be physical or abstract, stationary or moving. Note that certain areas of the surface (the pipe's walls) correspond to actual limits, whilst others (at positions CD, (2), and (3)) are fictitious areas of the surface (inlets or outputs). The flow rate at outlet (J) given the flow rates at inlet (T) and outlet (2) (similar to a problem we shall study in the force needed to keep the junction in place, etc.) might be calculated for the control volume defined by this surface using equations for the fundamental laws. Figure 1.2 represents fictitious areas of the surface of fluid (Figure 1.2).

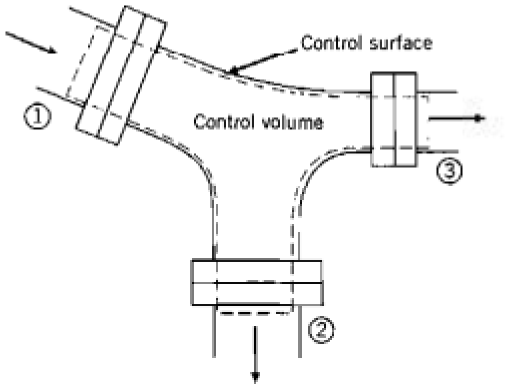

Figure 1.2 represents fictitious areas of the surface of fluid.

It goes without saying that the fundamental rules we'll be dealing with are the same ones employed in mechanics and thermodynamics. It will be our responsibility to develop these rules in a way that makes them useful for solving fluid flow issues and for a broad range of other scenarios. It is important to note that many seemingly straightforward issues in fluid mechanics cannot be analytically solved, as we will demonstrate. In these situations, we must turn to more complex numerical answers or the findings of experimental studies.

Dimensions and Units

Engineering issues are resolved to provide detailed answers to queries. Without a doubt, the solution must contain units. The engineers at JPL misread a measurement in 1999, causing NASA's Mars Pathfinder to crash because they thought it was in metres when it was really in feet! So it seems sense to provide a quick rundown of measurements and units. Because the subject is well-known from your past study in mechanics, we say "review."

Dimensions are the terms we use to describe the physical properties of length, time, mass, and temperature. All measurable quantities are split into two categories, main quantities and secondary quantities, according to a specific system of dimensions. We speak about a limited number of dimensions that may be used to create all other dimensions as fundamental quantities, for which we establish arbitrary scales of measurement. Secondary components are those quantities their dimensions may be expressed in terms of the main quantities' dimensions. The arbitrary names (and magnitudes) given to the fundamental dimensions used as measuring standards are known as units. The basic dimension of length, for instance, may be expressed in terms of metres, feet, miles, or miles. Through unit conversion factors, these length units are connected to one another (1 mile = 5280 feet = 1609 meters).

Dimensions System

Every term in an equation that connects physical quantities must have the same dimensions in order for the equation to be valid. We understand that the four dimensions of F, M, L, and t are related by Newton's second law (F * ma). Therefore, it is impossible to choose force and mass as the fundamental dimensions without adding a proportionality constant that has dimensions (and units).

In all commonly used dimensional systems, length and time are the two main dimensions. Mass is regarded as the main dimension in several systems. Others chose force as their fundamental dimension, while a third system choose both mass and force as its primary dimensions. As a result, there are three fundamental systems of dimensions that correspond to the many ways that the main dimensions might be specified.

1. Amount [M], Length [L], Duration [D], and Temperature [T].
2. b. Temperature [T], length [L], time [[/], and force [F].
3. Force [F], mass [M], length [L], time [] and temperature [T] are the other three.

Force [F] is a secondary dimension in system a, but Newton's second law's proportionality constant is a dimensionless constant. Mass [M] is a secondary dimension in system b, and Newton's second law's constant of proportionality is, once again, dimensionless. Force [F] and mass [M] have both been chosen as the major dimensions for system c. The proportionality constant, gc, in second law of motion (written F = md/gc), which is not to be confused with g, the acceleration of gravity!, is not dimensionless in this situation. The equation is dimensionally homogenous if and only if the dimensions of gc are [MLJFr2]. The units of measurement used for each of the fundamental variables affects the numerical value of the proportionality constant.

Applications of Fluid Mechanics

According to fluid mechanics. The pipe and ducting system of heating and air conditioning systems operates similarly. A refrigerator consists with the refrigerant runs via tubes, a compressor pressurizes the two different heat exchangers where refrigerator receives and rejects heat, and refrigerant heat. The design of each of these parts heavily draws on fluid dynamics. Even standard faucets need fluid mechanics to function.

The design and practical application of fluid mechanics are both common. of contemporary technical systems, ranging from supersonic aero planes to vacuum cleaners. Therefore, it's crucial to get a solid knowledge of fluid mechanics' foundational ideas. To start, fluid mechanics is important to the human body. The Blood is continuously pumped from the heart to all organs and portions of the body. The locations of airflow alternate between the lungs, arteries, and veins. Directions. It goes without saying that all artificial hearts, respiration apparatuses, and Fluid dynamics is used in the design of dialysis systems.

analyses of aero planes, ships, rockets, jet engines, and wind turbines, biomedical equipment, electronic component cooling, and the movement of liquids including water, crude oil, and fossil fuels. It is also taken into account when

Designing structures to handle wind loads includes bridges, billboards, and even skyscrapers. Several natural events, including the meteorological patterns, the ascent of groundwater sources to the tops of trees, the cycle of rainfall, large water bodies' currents, winds, and ocean waves are all controlled by the fundamentals of fluid dynamics.

When a material is over the critical temperature, it is commonly referred to be a gas. Vapor often denotes a gas that's also close to the condensation stage. Every actual fluid system is made up of a lot of molecules, and naturally, how these molecules behave affects the system's attributes. For instance, the outcome of a gas's pressure in a cylinder is momentum transfer here between molecule and the boundaries of the container. However, one does not necessarily need to understand how gas molecules behave to identify the container's pressure.

Fluid as a Continuum

The most prevalent ones are air and water, which humans perceive as being "smooth," or a continuous medium. We are unaware of the fundamental molecular makeup of fluids unless we utilise specialist equipment. This molecular structure has molecules that are spaced apart by relatively significant amounts of empty space, rather than having the mass constantly distributed across space. In this part, we'll talk about when a fluid may be thought of as a continuum, which by definition has features that gradually change from one point to the next.

The fundamental idea behind classical fluid mechanics is a continuum. The continuum concept is appropriate for handling fluid behaviour under typical circumstances. It only fails when the molecule's mean free path reaches the same order of magnitude as the problem's lowest meaningful characteristic dimension. This happens in complex issues like rarefied gas movement (e.g., as encountered in flights into the upper reaches of the atmosphere). We must give up the idea of a continuum for these particular circumstances (not treated in this chapter) in favour of the microscopic and analytical points of view.

The continuity assumption leads to the notion that every point in space has a specific value for every fluid attribute. As a result, fluid parameters like density, temperature, and velocity are thought of as continuous functions of location and time.

One-, Two-, and Three-Dimensional Flows

Depending on the number of spatial coordinates necessary to define the velocity field, a flow is characterised as one-, two-, or three-dimensional.

According to Equation, the flow field may be modelled as a function involving three spatial coordinates and time. A three-dimensional flow field (which is also unstable) is so named because the velocity at every location in the flow field relies on the three coordinates necessary to identify the place in space.

Although most flow fields are intrinsically three-dimensional, analysis based on fewer dimensions is typically useful. Consider the constant flow through a long pipe diameter with a diverging portion, in this example, we'll use cylindrical measurements (r, 9, x) under some conditions (for example, far from the pipe's entry and from the divergent portion, where the flow might be extremely convoluted), the velocity distribution can be characterised by:

$$u = u_{\max}\left[1-\left(\frac{r}{R}\right)^{2}\right]$$

Because the velocity u(r) is solely a function of one coordinate, the flow is one-dimensional. In the diverging portion, however, the velocity drops in the jr-direction and the flow becomes two-dimensional: u = ur, x), the amount of dimensions in the flow field raises the analysis's complexity significantly. For many engineering issues, a one-dimensional analysis is sufficient to produce engineering-accurate solutions. Most flows are intrinsically two- or three-dimensional since all fluids meeting the continuum assumption must have zero angular velocity at a hard surface (to satisfy the no-slip criterion). It is frequently advantageous to utilize the concept of uniform flow at a particular cross section to simplify the analysis. The velocity in a uniform flow at a particular cross section is constant throughout any portion equal to the flow. Under this assumption, the two-dimensional flow is represented as the Figure 1.3 Represents the One-, Two-, and Three-Dimensional Flows.

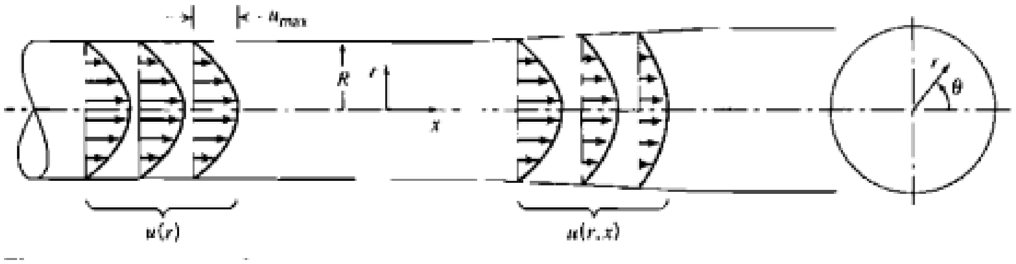

Figure 1.3 Represents the One-, Two-, and Three-Dimensional Flows.

Stress Field

Stress will need to grasp what kind of forces operate on fluid particles in fluid mechanics. Surface forces (pressure, friction) created by contact with some other particles or a solid surface may be experienced by each fluid particle, as can body forces (such as gravity and electromagnetism) experienced throughout the particle. The gravitational body force acting on a volume element, dV, is given by p gdV, where p denotes density (mass per unit volume) and g denotes local gravitational acceleration. Thus, pg denotes the weighted body force per unit volume, while sg denotes the gravitational body force per unit mass.

Stresses are caused by surface forces on a fluid particle. The idea of stress is important for defining how forces acting on a medium's (fluid or solid) boundaries are propagated across the medium. You've undoubtedly heard of stresses in solid mechanics. When you stand on a diving board, for example, tensions are created inside the board. When a body travels through a fluid, however, tensions emerge inside the fluid. As we've seen, the distinction between such a fluid and a solid is that stresses in a fluid are largely created by motion rather than deflection. Consider the surface of a fluid particle in touch with another fluid particle, and the contact force created between the particles. Consider a section of the surface, 8 A, at some point C. The vector h represents the particle's externally drawn unit normal. The force operating on 8A, 8F, may be divided into two components, one normal to the area and the other perpendicular to the area. Then a normal stress crn and a shear stress T,, are defined.

CHAPTER 2

CLASSIFICATION OF FLUID MOTIONS

Mr. Alok Raj,
Assistant Professor, Department of Chemical Engineering,
School of Engineering & Technology, Jaipur National University, Jaipur, India,
Email Id-aloksraj06@jnujaipur.ac.in

Fluid mechanics is a vast subject that includes everything from the aerodynamic of a supersonic transport aircraft to the lubricating of human joints with sinovial fluid. Fluid mechanics must be broken down into digestible chunks. The two most challenging parts of a fluid mechanics study to deal with are: (the viscous nature of the fluid and its differential pressure. In fact, dealing with either a frictionless, incompressible fluid was the first area of continuum mechanics theory to become well developed (approximately 250 years ago!). As we shall see soon and in further detail later, this theory, although incredibly elegant, resulted in the famous d'Alembert's paradox: When moving through such a fluid, all bodies feel no drag a conclusion that is inconsistent with any genuine behavior.

Viscous and Inviscid Flows

When the viscosity is extremely low or virtually nonexistent, i.e. there is no friction between the fluid layers, the flow is believed to be inviscid. There is also little or no mass redistribution or heat conduction between the particles. Because Reynolds number is inversely related to viscosity, which approaches 0 for inviscid flow, it approaches infinity. Because viscosity is zero or unimportant, the Navier-Stokes equation may be further reduced to the Euler equation. Although there is no true inviscid flow in nature, we use it to simplify calculations for Mach numbers less than 0.3 since Bernoulli's principle can then be used, which seems to produce accurate results. Shear stress acting throughout the body will be very minimal, if not nonexistent. Because there are no net two forces that affect the body in the inviscid flow, no aerodynamic forces are created. As a result, inviscid flow has no lift or drag. The flow lines are symmetric and parallel to the streamline flow direction. The pressure is spread symmetrically over the body's surface. The acceleration at the body's surface will not be equal to zero. The velocity at the body's surface will be zero in viscous flow. There will be a differential in pressure distribution and shear stress throughout the body due to the existence of viscosity. As a result, the net imbalanced forces will be greater than zero, and the body will generate air flow such as lift and drag. Because velocity varies along the streamlines, pressure cannot be distributed symmetrically throughout the body's surface.

Laminar and Turbulent Flows

In pipes (or tubes), laminar flow or streamline flow occurs when one fluid flows in parallel layers with no disturbance between them. The fluid typically flows without lateral mixing at low velocities, and neighbouring layers glide past one another like playing cards. There are no crosscurrents perpendicular to the flow direction, nor are there any eddies or swirls of fluid. The motion of the fluid particles in laminar flow is exceedingly ordered, with all particles travelling in straight lines along to the pipe walls. The process of diffusion between liquid layers causes any lateral mixing (mixing at right angles to the flow direction). Diffusion mixing may be sluggish, but if the pipe diameter of the tube is tiny, this diffusive mix can be quite substantial.

Turbulent flow is characterised by erratic property changes in the flow. This comprises a fast change in pressure and flow velocity throughout time and place. Unlike with laminar flow, the fluid does not move in layers, and mixing throughout the tube is very efficient. Flows with Reynolds numbers more than 4000 are frequently (but not always) turbulent, while flows with Reynolds numbers less than 2300 are normally laminar. Transition flow is defined as flow with Reynolds numbers ranging from 2300 to 4000.

When operated at different flow rates, laminar and turbulent flows may coexist in the same tube network. A Vapourtec 1mm bore tubular reactor running water at 10 ml/min has a Reynolds number just over 200. We may fairly presume that the flow through the tube reactors of commercial flow chemistry systems is Laminar Flow under typical working circumstances. Turbulent flow is a typical event that may be seen in the flow of rivers, smoke flowing out of tail pipes, or even experienced during aeroplane turbulence. This turbulent effect is caused by an irregularity or blockage in the fluid flow route. Unlike in laminar flow, the fluid layers in shear stress may cross pathways as the amplitude and direction of the flow vary continuously. In turbulent flow, Eddies or swirls may be seen, and the seeming irregular flow behaviour makes turbulent flow analysis difficult. Despite the difficulties, turbulent flow analysis is vital for companies since most flows seen are turbulent. Turbulence analysis may aid in the successful design of liquid viscosity or mixing systems, the examination of buildings such as bridges or wind tunnels, and the design of fuel-efficient automobiles and aeroplanes in the automotive industry. Reynolds number may be used to forecast turbulent flow to some degree.

Using Reynolds Number to Determine Flow Pattern

1. where is the fluid's density
2. V denotes the fluid velocity
3. D denotes the droplet size (of pipe, tube, or duct)
4. is the viscosity of the fluid

The following are important insights from the Prandtl number calculation:

1. If the Reynolds number is more than 2300, the flow is termed laminar. Because of the slower flow rate, viscous force is more pronounced.
2. If the Reynolds index is more than 3500, the flow is called turbulent. A quicker and more irregular flow channel boosts the system's inertial force. Transitional flow is defined as a flow regime with a Reynolds number ranging from 2300 to 3500.

Compressible and Incompressible Flows

Compressible fluids are gases (including plasma=ionized gas) in general. The volume or density of a fluid does not vary under normal temperature and pressure circumstances. However, even little changes in temperature in pressure cause a change in volume (and hence a change in density). To be called a specific fluid compressible, it must exhibit a significant change in density when pressure or force is applied.

In more sophisticated fluid dynamics jargon, the ratio of flow velocity to sound velocity in a fluid is larger than 0.3 for incompressible flows. This ratio is also known as the Mach number.

When pressure is applied to a gas at the molecular level, the pressure impacts the gas in all directions, resulting in a high degree of collisions between the molecules of the gas. These collisions allow the gas molecules more time to interact with one another, and greater repulsion forces between molecules may arise. The velocity of gas molecules is reduced by these attraction forces. As a consequence, the gas is compressed.

Incompressible fluids are liquids. When pressure is applied to a liquid, its volume or density does not alter readily. For a fluid to be incompressible, the ratio in flow velocity and sound velocity in the medium should be less than 0.3, according to fluid dynamics. As a result, this ratio is less to 0.3 for liquids, indicating that it is an incompressible fluid. The molecules or atoms of liquids are more densely packed than those of gases As a result, applying pressure on a liquid has little effect on its density. In other words, pushing down on the liquid does not decrease its volume. Although fluid are considered incompressible by fluid dynamics, they are compressible when pressure is applied but really the change UN density or area is too tiny to determine. As a result, it is classified as an incompressible flow.

Internal and External Flows

Internal flow is a kind of flow in fluid dynamics in which the fluid is contained by a surface. Because circular pipes can bear high pressures and are therefore utilised to transfer liquids, detailed understanding of the behaviour of interior flow regimes is essential in engineering. Non-circular ducts are utilised in heating and air conditioning systems to transmit low-pressure gases such as air.

Internal flow geometry is a useful geometry for heating and cooling liquid used in technologies for energy conversion such as nuclear power plants. External flow is defined in fluid dynamics as a flow for which boundary layers form freely, without being constrained by nearby surfaces. External flows, in contrast to internal flows, have very viscous effects that are restricted to quickly expanding "border layers" near the entry area or thin shear levels along the solid surface. As a result, there will always be a flow zone beyond the boundary layer. Velocity, temperature, and/or density do not fluctuate in this area, and their gradients may be ignored.

This action causes the flow separation to expand, and the thickness of the boundary layer is proportional to the fluid's kinematic viscosity.

This is seen in the following image. The flow is practically invisible away from the body. It is described as the movement of fluids around a totally submerged body in it. Figure 2.1 Internal and External Flows.

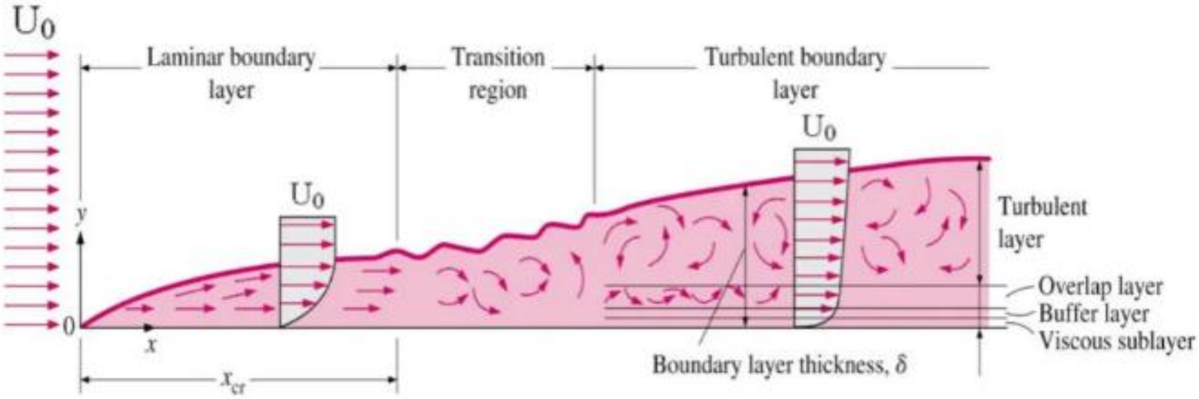

Figure 2.1 Internal and External Flows.

Fluid Properties and Hydrostatics in Brief

Density

The density of a substance indicates the density of that material in a specified region. The density of a substance is defined as its mass per unit volume. Density is a measurement of how closely stuff is packed together. It is a distinct physical attribute of a certain thing. Archimedes, a Greek physicist, developed the density principle. If you know the formula and understand the units, calculating density is simple. Density is symbolised by the symbol, which may alternatively be represented by the letter D.

Density Examples

Dense materials include ron, platinum, and lead. Dense material includes several different forms of rock and minerals. Dense materials are more likely to 'feel' weighty or hard. Sparse is the inverse of dense, and some examples of sparse materials are glass, bamboo, aluminium, and fibreglass. Liquids are generally less dense than solids, while gases are less opaque than liquids. This is because solids contain closely packed particles, liquids have particles that can slide round one another, and gases contain particles which are free to move about.

Unit of Density

Though the SI unit of density is kg/m3, we use g/cm3 for solids, g/ml for liquids, and g/L for gases for convenience.Density is defined as the connection between a substance's mass and the volume it occupies. In a qualitative sense, it indicates how weighty an item is at constant volume. Varying substances have different volumes, which implies they weigh differently for the same volume.

Applications of Density in Real Life

There are several uses of density in everyday life, including pipe engineering, shipbuilding, helium balloons, weight distribution in aeroplanes, and the fact that ice floats on water. Understanding the densities of two liquids aids you in separation strategies. As an example, consider the separation oil oil from water. If an oil tank leaks in the ocean, oil drips begin to float on the sea owing to their lower density than water. Another well-known use of density is deciding whether or not an item will float on water. The difference in density causes ships to float and submarines to dive.

Viscosity

The resistance of a fluid (liquid or gas) to a change in form or movement of neighbouring sections relative to one another is referred to as viscosity. Viscosity signifies resistance to flow. Fluidity is an index of the ease of flow that is equal to the reciprocal of viscosity. Molasses, for example, is more viscous than water. Because a moving component of a fluid carries along neighbouring sections to some degree, viscosity may be thought of as compressive stress between molecules; this friction resists the formation of velocity inequalities within a fluid. When fluids are employed in lubrication and carried in pipelines, viscosity is a crucial element in determining the forces that must be overcome. It regulates liquid flow in operations like as spraying, injection moulding, and surface coating.

The tangential, or shearing, stress that generates flow in many fluids is exactly proportional to the rate of shear strain, or rate of deformation. In other words, for a given fluid at a certain temperature, the compressive force divided by the rate of shear strain is constant. This constant is referred to as the dynamic, or absolute, viscosity, or simply the viscosity. Fluids that act in this manner are known as Newtonian fluids, after Isaac Newton, who originally developed this mathematical explanation of viscosity.

The viscosity of liquids drops fast as temperature rises, but the viscosity of gases rises as temperature rises. As a result, liquids flow more freely when heated, and gases flow more slowly. Water viscosities at 27 °C (81 °F) and 77 °C (171 °F) are 0.85 103 and 0.36 103 pascal-second, respectively, but air viscosities at the same temperatures are 1.85 105 and 2.08 105 pascal-second.

Kinematic viscosity is more helpful than absolute, or dynamic, viscosity in certain situations. Kinematic viscosity is defined as a fluid's absolute viscosity divided by its mass density. (Mass density is defined as a substance's mass divided by its volume.) Kinematic viscosity is defined as area divided by time; the suitable units are metre squared per second. The centimetre-gram-second (CGS) system's unit of kinematic viscosity, known as stokes in the United Kingdom and stoke in the United States, is named after British scientist Sir George Gabriel Stokes. One stroke is one cm squared per second.

Surface tension

Surface tension is a feature of a liquid surface that manifests itself as if it were a stretched elastic membrane. This phenomenon may be seen in the roughly spherical form of tiny liquid droplets and soap bubbles. Certain insects may float on the surface of water due to this feature. The surface tension of water may also sustain a razor blade. When pushed through the surface, the razor blade sinks into the water. Surface tension is primarily determined by the forces of attraction among the particles in a specific liquid as well as the gas, solid, or liquid in contact with it. The molecules in a drop of water, for example, have a modest attraction to one another. Water molecules deep inside the drop may be conceived of as being equally drawn in all directions either by surrounding molecules. However, if surface molecules were shifted slightly away from the surface, they would be repelled by neighboring molecules. Surface tension energy may be thought of as being roughly similar to the effort or energy necessary to remove the surface layer of molecules in a square meter. Surface tension may therefore be described in terms of energy (joules) per unit surface area (square metres). At 20 °C (68 °F), water has a surface tension of 0.07275 joule per square metre. Organic liquids, such as benzene and alcohols, have low specific tensions than mercury, which has a greater surface tension. As the temperature rises, the net attraction forces between molecules diminishes, as does surface tension.

Compressibility

The compressibility (also called the coefficient of compressibility or, if the temperature is maintained constant, the isothermal compressibility is a measure of a fluid's or solid's instantaneous relative volume change in response to a pressure (or mean stress) rise. The compressibility (denoted in several domains) may be represented simply as

$$\beta = -\frac{1}{V}\frac{\partial V}{\partial p},$$

Where V denotes volume and p denotes pressure. Because compressibility is defined as the inverse of the percentage, it is positive in the (typical) scenario when a rise in pressure causes a decrease in volume. The isothermal bulk modulus is defined as the reciprocal of compressibility at a constant temperature.

Compressibility is the amount by which a given volume of substance reduces when subjected to pressure. When we apply pressure to a solid or a liquid, there is almost little change in volume. The atoms, ion, or molecules that comprise the solid or liquid are very near. Because there is no room between the particles, they cannot compact together.

Gases are more compressible than liquids or solids, according to the kinetic-molecular hypothesis. Gases are compressible because the majority of their volume is made up of enormous quantities of empty space between gas particles. The average distance between gas molecules at normal temperature and standard pressure is around 10 times the circumference of the molecules themselves. When a gas is compressed, such as when filling a scuba tank, the gas particles are squeezed closer together. Compressed gases are employed in a variety of applications. In hospitals, oxygen is often used to assist patients with damaged lungs breathe better. When a patient has a major procedure, the anaesthetic used is often a compressed gas. Welding requires very hot sparks generated by compressed acetylene and oxygen mixtures. Compressed propane is used to power many summer barbecues.

Capillarity

Capillarity is the rise or descent of a liquid in a narrow route, such as a tube with a small cross-sectional area, such as the gaps between towel fibres or pores in a porous substance. Capillarity does not just exist in the vertical direction. Water is pulled into the strands of a towel regardless of how it is orientated.

Liquids rising in small-bore tubes put into the liquid are said to wet the tube, whilst liquids depressed inside thin tubes below the surface of surrounding liquid are said not to wet the tube. Water is a liquid than wets glass capillary tubes whereas mercury does not. Capillarity does not exist when there is no wetness. Surface, or interfacial, forces cause capillarity. Water rises in a thin tube immersed in water due to forces of attraction between the molecule of water and the glass walls, as well as between the molecules of water themselves. These attractive forces simply balance the gravitational force of the column of water that has risen to a certain height. The higher the water rises, the smaller the diameter of the capillary tube. Mercury, on the other hand, is depressed to a larger extent as the bore narrows.

Vapour Pressure and Cavitation

When a liquid is drawn into a duct, it causes a pressure drop. If the fluid pressure falls below the saturation vapour pressure, the liquid starts to boil. This phenomena is known as cavitation in hydraulic (steam generation). The development of vapour bubbles as a result of pressure loss is known as cavitation. At a given temperature, the saturation vapour pressure is the pressure at which a fluid transitions from the gaseous to the liquid state (or from liquid to gas). The pressure at which the fluid transforms from liquid to gas (saturated vapour pressure) increases as the temperature of the fluid rises. Thus, a liquid, such as water, may be converted to vapour at ambient pressure by applying heat; however, this transformation can be accomplished without altering the temperature by reducing the pressure gradient below the saturated vapour pressure. When a liquid is drawn into a duct, it causes a pressure drop. If the fluid pressure falls below the saturation vapour pressure, the liquid starts to boil. This phenomena is known as cavitation in hydraulic (steam generation).

The development of vapour bubbles as a result of pressure loss is known as cavitation. The formation of bubbles increases the amount of fluid in the low differential pressure, which raises pressure in certain locations where the gas bubble condenses forcefully, bursting. The shocks produced by bursting bubbles damage the organ walls that come into touch with the fluid. A cavitation pump wears out rapidly.

$$\ln \frac{P_{sat}}{P_0} = \frac{M.L_v}{R}(\frac{1}{T_0} - \frac{1}{T})$$

Hydrostatic forces

Hydrostatic forces are the consequence of a liquid's pressure loading acting on buried surfaces. Fluid mechanics begins with the calculation of the hydrostatic force as well as the position of the centre of pressure.

Inclined and Curved Surfaces

When an item is put on a slanted surface, it often slides down the surface. The velocity at which the item slides down the surface is determined by its tilt; the higher the tilt of the ground, the quicker the object will slide down it. An inclined plane is a physics term for a slanted surface. An imbalanced force causes objects to accelerate down sloped planes. To comprehend this form of motion, consider the forces operating on an item on an inclined plane. The right-hand figure displays the two forces acting on a box positioned on a plane surface (assumed to be friction-free). As seen in the picture, every item positioned on a plane surface is always subject to at least two forces: gravity and normal force.

The Abnormal Normal Force

The primary feature of inclined plane issues is that the force f is not applied in the usual direction. We have always observed normal forces working in an upward direction, against the direction of gravity up to this point in the course. This is due to the fact that the items were always on sloping ground and never on inclined planes. The reality about normal forces is that they are always directed toward the surface that the item is on, not necessarily upwards. Figure 2.2 Abnormal Normal Force.

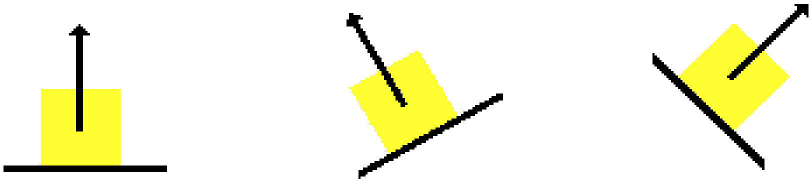

Figure 2.2 Abnormal Normal Force.

The Components of the Gravity Force

Because the two (or more) impulses are not oriented in opposing directions, finding the net force exerted on an item on an inclined plane is a tough operation. As a result, one (or more) of the forces must be resolved into perpendicular components that can readily be added to the other forces operating on the object. Any motion controller at an angle to the horizontal is usually decomposed into horizontal and vertical elements. However, this is not the approach we shall use with inclined planes. Instead, the weight vector (Fgrav) will be divided into two perpendicular components in order to analyse the forces acting on objects on inclined planes. This is the second distinguishing feature of inclined plane issues. The force of gravity will be divided into two components, one parallel to the inclined surface and one perpendicular to the incline plane. The figure below depicts how gravity has been replaced by two components of force: a parallel and a perpendicular element of force. Figure 2.3 Components Of The Gravity Force.

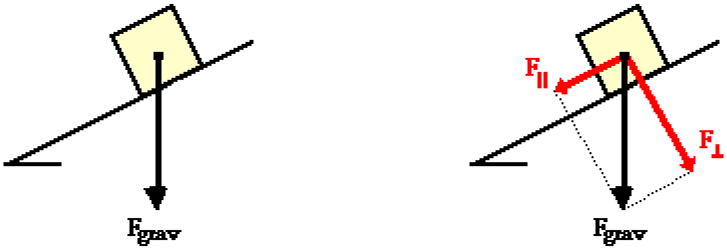

Figure 2.3 Components Of The Gravity Force.

The perpendicular component of gravity's force is directed towards the normal force and thereby balances it. There is no other force that can balance the parallel component of gravity. Due to the existence of an imbalanced force, this item will then accelerate down the airfoil. This acceleration is caused by the parallel component of gravity's force. The net force is the perpendicular component of gravity's force.

The issue becomes significantly more difficult in the presence of friction or other forces (applied force, tectonic forces, etc.). Take a look at the diagram on the right. Because objects do not accelerate perpendicular to the slope, the perpendicular component of force still matches the normal force. However, while calculating the net force, the frictional force must also be included. The net force, like in other net force issues, is the vector of all the forces. In other words, all of the separate forces are totaled together as vectors. The sum of the perpendicular component and the normal force is 0 N. The parallel element and the friction force add up to 5 N. The net force is 5 N, directed towards the floor down the slope.

The aforementioned issue (and other inclined plane problems) may be reduced by using a technique known as "tilting the head." An inclined plane issue is identical to any other net force question except that the surface has been angled. Simply tilt your head in the same way as the inclination was slanted to turn the issue back into the shape with that you feel more comfortable. Better still, just tilt the piece of paper so that the surface no longer seems level (a guaranteed cure for TNS - "tilted neck syndrome" / "taco neck syndrome"). This is shown below. Figure 2.4 aforementioned issue (and other inclined plane problems).

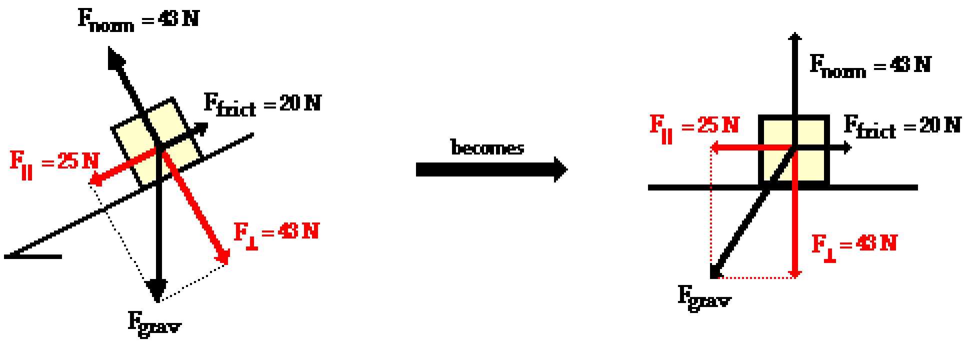

Figure 2.4 aforementioned issue (and other inclined plane problems).

The issue should appear fairly familiar after the gravity's force has been resolved into its two components and the sloping plane has been slanted. Simply disregard gravity's force (which has been superseded by its two components) and solve for net force and acceleration. Consider the scenario represented in the figure to the right. The forces operating on a 100-kg container sliding down an air track are shown in the free-body diagram. The plane is sloped at a 30 degree angle. The friction coefficient between the container and the hill is 0.3. Determine the crate's net force and acceleration.

Begin by determining the force of gravity acting on the container, as well as the components of this force horizontal and perpendicular to the slope. Gravity's force is 980 N, and its components are Fparallel = 490 N (980 N * sin 30 degrees) and Fperpendicular = 849 N (980 N * cos30 degrees). The normal force may now be calculated to be 849 N. (it must balance the perpendicular component of the weight vector). The friction force may be calculated using the normal force and the coefficient of friction; Ffrict is 255 N (Ffrict = "mu"*Fnorm= 0.3 *849 N). The vector sum of all forces is the net force. The forces perpendicular to the inclination balance; the forces parallel to the incline do not balance. The total net force is 235 N. (490 N - 255 N). 2.35 m/s/s (Fnet/m = 235 N/100 kg) is the acceleration.

Some Roller Coaster Physics

Roller coasters provide two thrills, the first of which is linked with the first plunge down a steep slope. The thrill of acceleration is provided by employing steep inclination angles on the initial descent; such big angles enhance the value of the weight vector's parallel component (the component that causes acceleration). Weightlessness is achieved by lowering the magnitude of normal force to levels smaller than their typical values. It is critical to understand that the sensation of weightlessness is connected with a lower than typical normal force. When sitting on a chair, a person weight 700 N will typically feel a 700 N normal force. If, on the other hand, the chair is moving down a 60-degree slope, the user will feel a 350 Newton normal force. This number is lower than usual, which leads to the sensation of weighing much less one's normal weight, i.e. weightlessness.

Archimedes' Principle

Archimedes' principle is a physical law of buoyancy discovered by the ancient Greek mathematician and inventor Archimedes, which states that anybody whole or submerged in a fluid (gas or liquid) at rest is acted by a upward, or buoyant, force whose magnitude is equal to the weight of the fluid displaced by the body. The volume of displaced fluid is equal to the volume of a completely immersed item in a fluid or to a proportion of the volume just below surface of a partly submerged object in a liquid. The weight of the displaced fluid is proportional to the size of the buoyant force. The buoyant force applied to a body floating in a liquid or gas is similarly comparable in size to the floating item's weight and is directed in the opposite direction; the object does not rise or sink. A ship, for example, sinks into the ocean until the weight of the liquid it displaces equals its own weight. As the ship is loaded, it lowers deeper, displacing more water, and the magnitude of the buoyant force continues to match the ship's and cargo's weight.

If an item's weight is less than that of the displaced liquid, the piece rises, as in the instance of a piece of wood dropped under the surface of water or a helium-filled balloon discharged into the air. Though an item heavier than the quantity of fluid expelled sinks when released, the apparent weight loss is equal to the weight of the fluid displaced. In reality, some precise weighings need an adjustment to account for the buoyancy impact of the surrounding atmosphere.

Assume a basin is completely filled with water. What happens to the level of water when an aluminium foil barge is put on it? The barge is claimed to be buoyant at this stage. But what would happen if many coins were placed to the barge?

Will the barge remain afloat, or will it sink? In this case, the barge employs the idea of buoyancy. What exactly does the term "buoyant" mean? What is the definition of buoyant force? In the next sections, you will learn more about buoyancy definition, buoyant force, and its applications.

Buoyancy is the propensity of an item to float in any fluid. The buoyant force keeps objects floating in fluids. It is an upward force that the liquid exerts on the submerged object. Buoyant force is shown by a ship floating in the middle of the sea, an anchor sinking when dropped in the water, and even a fish hovering in the centre.

The idea of buoyancy was developed by Archimedes (287-212 B.C.), a Greek mathematician, scientist, and astronomer. His famous Eureka! Moment came when he was sitting in a tub, contemplating how to verify whether King Hieron II of Syracuse's crown is really composed of gold or has been mingled with silver. He then solved the problem by comparing the amount of water displaced by the king's crown to the volume displaced by a bar of gold of the same mass. He noticed that the crown had been tampered with and that the jeweller had really defrauded the monarch.

Archimedes' Principle Derivation

We already know that density is defined as

$$Density\,(\rho) = \frac{Mass\,(M)}{Volume\,(V)}$$

As a result, the displaced liquid's mass may be expressed as follows:

$$Mass\,(M) = Density\,(\rho) Volume\,(V)$$

The displaced liquid's weight may now be computed as follows:

$$Weight = Mass \times Acceleration\,due\,to\,gravity$$

$$Weight = Mass \times g = \rho \times V \times g$$

We know from Archimedes' principle that the apparent loss of weight equals the weight of water displaced, hence the thrust force is given by the following equation:

$$Thrust\,Force = \rho \times V \times g$$

Where the density of the liquid, V is the volume of liquid displaced, and g is gravity's acceleration. Because it is responsible for items floating, the thrust force is also known as the buoyant force. As a result, this solution is also known as the law of buoyancy.

Buoyancy

The propensity of an item to float in a fluid is referred to as buoyancy. In the presence of gravity, all liquids and gases exert an upward push known as the buoyant force on any object submerged in them. Buoyancy is caused by pressure differences acting on opposing sides of an item submerged in a static fluid. You will be able to understand the buoyant force why and fluids exert an upward surface tension on submerged objects after reading this article. Let's go through the definition of buoyancy and the many characteristics of buoyant force.

Buoyant Force

The buoyant force is upward and force exerted on an object submerged entirely or partially in a fluid. Upthrust is another name for this upward force. A body immersed partly or completely in a fluid looks to lose weight, i.e. appears to be lighter, due to buoyant force.

1. The buoyant force is affected by the following factors:

2. The liquid's density

3. The volume of both the fluid displaced the local gravitational acceleration

An item that has a higher density than the fluid in which it is immersed sinks. The force may keep an item afloat if it is less if dense than the liquid is either shaped suitably (as in a boat). In terms of relative density, compounds having a relative density less than one float in water, whereas substances with such a relative density larger than one sink.

Causes of Buoyant Force

When we submerge an item in water or another fluid, we notice that it receives a downward force opposing to the force of gravity, which is fundamental for the object's weight drop. The fluid's upward force resists the weight of an item submerged in fluid. The density in a fluid column, as we know, rises with depth. As a result, the pressure at the bottom of an item immersed in fluid is larger than the pressure at the top. The difference in pressure causes a net upward pull on the item, which we call buoyancy. Figure 2.5 Buoyant Force.

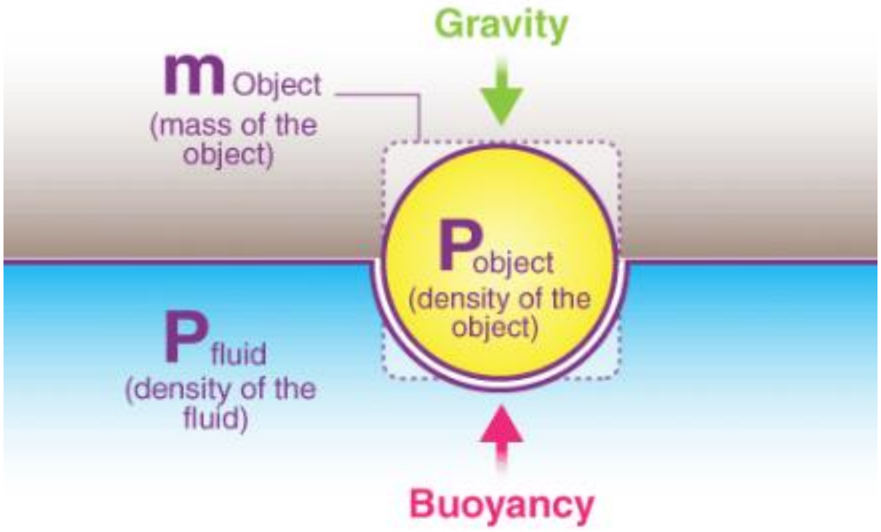

Figure 2.5 Buoyant Force.

When we immerse an item in a fluid, it experiences an upward force. The fluid exerts this force upon that item, causing it to rise; this force is known as buoyant force. The intensity of this force is exactly equal to the quantity of liquid displaced. The center of buoyancy is the place where the buoyancy force is applied or the location on the object where the force operates (Figure 2.6).

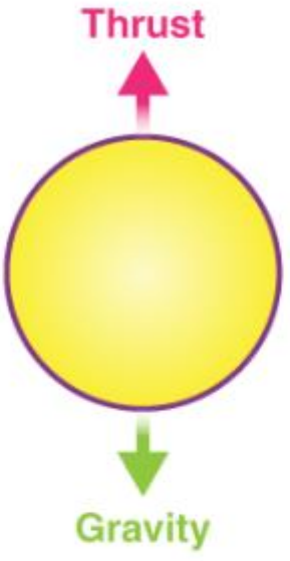

Figure 2.6 Thrust and gravity position enrolled in Buoyancy.

Object Float or Sink in Water

A single column of liquid may be thought of as a mixture of multiple overlapping layers, one on top of the other, with varying pressure. The pressure at the bottom of the liquid would be larger than at the top because the number of layers of the liquid that are overlaying, i.e. the layers one over the other, rises as we move down in the liquid.

Because of the difference in pressure between the layers, a made-up force is imparted on it in the upward direction. This force causes the item that has been submerged to accelerate upward. The impulse is always directed vertically. The amount of the upward force is also comparable to the difference in pressure between the topmost and last layers, as well as the quantity of the fluid expelled. Floating is the result of the preceding notion. The item should be less dense than water because otherwise, it will sink if its density is larger.

Positive, Negative, and Neutral Buoyancy

Whether completely or partly submerged, the buoyant force operates on all things. But how can one know if an item will swim or sink in the fluid. This is when the various forms of buoyancy (positive, positive, and neutral) come into play.

Positive buoyancy occurs when the weight of the displaced fluid exceeds the weight of the item, enabling it to float. This indicates that the upward force is much stronger than the downward force, resulting in a positive net buoyant force. Positive buoyancy is required for boats and ships to remain afloat.

Negative buoyancy occurs when the fluid load displaced is less than the weight of the item. Because the downward force from the item's weight exceeds the upward force, the net buoyancy force is negative, and the object sinks.

Metacentre

In fluid mechanics, the metacentre, also spelled metacenter, is the theoretical point at which an imaginary vertical line passing through the centre of buoyancy and gravity intersects the imaginary vertical line passing through a new centre of buoyancy created when the body is displaced, or tipped, in the water, however slightly.

A floating body's centre of buoyancy is the point at which all of its pieces precisely buoy one another and other words, the effective core of the displaced water. Regardless of the tilt of a floating body, such as a ship, the metacentre stays immediately above the centre of buoyancy. When the vessel is at rest on an even keel, the centre of buoyancy lies immediately below the centre of gravity as well as below the metacentre. The centre of gravity is the point in a body around which all parts of the body balance each other. When a warship tilts, one side displaces more water than the other, and the centre of buoyancy moves and is no longer directly under the centre of gravity; however, regardless of the amount of tilt, the centre of buoyancy remains directly below the metacentre. When the ship tilts, buoyancy provides stability if the metacentre is higher than the centre of gravity. The distance between the metacenter and the centre of gravity, known as the metacentric height, enhances the stability. The boat is unstable if the metacentre is lower than the centre of gravity, and a tilt results with capsizing.

Hydraulics

Hydraulics is a discipline of science dealing with the practical uses of fluids in motion, mainly liquids. It is connected to fluid mechanics, which provides most of its theoretical underpinning. Hydraulics is concerned with the movement of liquids through pipelines, rivers, and channels, as well as its restriction by dams and tanks. Some of its concepts also apply to gases, generally in circumstances when density fluctuations are minimal. As a result, the scope of hydraulics includes mechanical devices such as fans and gas turbines, as well as pneumatic control systems.

For many millennia, liquids in motion or under pressure performed beneficial work for mankind before French scientist-philosopher Blaise Pascal and Swiss physicist Daniel Bernoulli developed the rules that underpin contemporary hydraulic power technology. Pascal's principle, developed about 1650, says that pressure in a liquid is transferred equally in all directions; that is, when water is forced into a closed container, pressure applied at any point is communicated to all sides of the container. A little force supplied to a small piston in a small cylinder is conveyed via a tube to a big cylinder, where it pushes evenly against all sides of the cylinder, including the huge piston, to obtain an increase in force. Bernoulli's theorem, developed about a century later, argues that energy in a fluid is related to elevation, velocity, and pressure, and that if there are no friction losses and no work done, the total of the energies stays constant. Thus, kinetic energy may be partially converted to pressure energy by increasing the cross section of a pipe, which slows the flow but increases the area against which the fluid is pushing.

Until the nineteenth century, it was impossible to create velocities and pressures considerably higher than those given by nature, but the advent of pumps opened up a large possibility for applying Pascal and Bernoulli's discoveries. In 1882, the city of London constructed a hydraulic system that fed pressured water via street mains to power factory machines. When an oil hydraulic system was constructed to lift and regulate the guns of the USS Virginia in 1906, it was a significant development in hydraulic methods. Self-contained hydraulic systems with a pump, controller, and motor were invented in the 1920s, paving the way for uses in machine tools, vehicles, agricultural equipment, earth-moving gear, locomotives, ships, aircraft, and spacecraft.

There are five components in hydraulic power systems: the driver, the pump, the control valves, the motor, and the load. The driver might be an electric motor or any form of engine. The pump's primary function is to raise pressure. The motor is similar to the pump in that it converts hydraulic input into mechanical output. Motors may cause rotational or reciprocating motion in the load. Fluid power may compete effectively with mechanical and electrical systems in the operation and control of machine tools, agricultural equipment, construction machinery, and mining machinery (see fluidics). Its primary benefits are flexibility and the capacity to effectively increase forces; it also delivers rapid and precise response to controls.

Hydraulic power systems have evolved into one of the most important energy-transmission technologies employed in all aspects of industrial, agriculture, and military activities. Hydraulic systems, for example, are used in modern aeroplanes to activate controls as well as to operate landing gears and brakes. Fluid power is used by almost all missiles and ground-support systems. Hydraulic power systems are used in automobile gearboxes, brakes, and steering mechanisms. In many sectors, mass production and its progeny, automation, have their roots in the usage of hydraulic systems. Fracking, or hydraulic fracturing, has enabled the production of petroleum and natural gas form previously unreachable resources.

CHAPTER 3

PRESSURE AND ITS MEASUREMENT

Dr. Avdesh Singh Pundir,
Associate Professor, Department of Chemical Engineering, Jaipur National University, Jaipur
India
Email Id-dr.avdeshpundir@jnujaipur.ac.in

The pressure at a place inside a fluid caused by the fluid's weight is known as fluid pressure. Furthermore, fluid pressure amplification may occur through hydraulic processes and changes in fluid velocity. Take the example of pressure difference in a column. Furthermore, as the depth of a fluid column grows, so does the pressure. Most notably, the reason for this pressure rise is that as one travels deeper, fluid at a lower level must support fluid above it. Figure 3.1 Pressure and its measurement

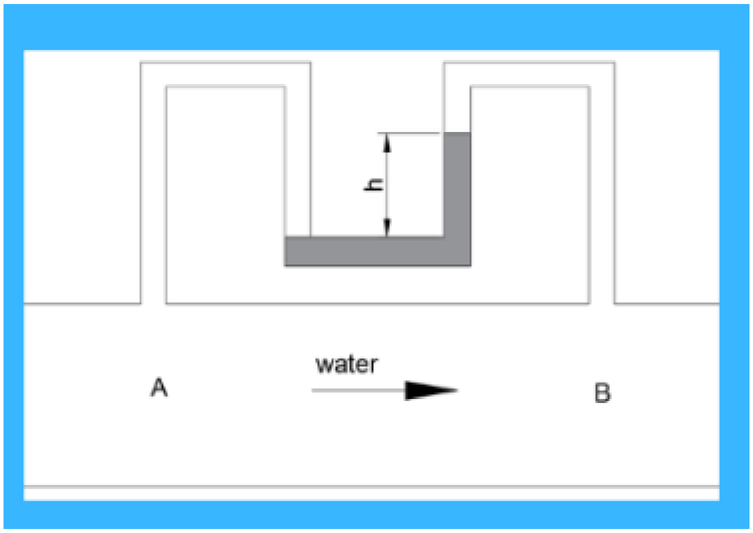

Figure 3.1 Pressure and its measurement.

Measure of Fluid Pressure At a point

When a fluid is at rest, it exerts a force perpendicular to every surface with which it comes into touch. This is known as fluid pressure, and it is caused by the random movement of molecules. Furthermore, fluid pressure measurements are made in Pascals (Pa).

It's worth noting that one Pascal equals one Newton for square metre (N/m2). Furthermore, fluid pressure is not affected by the mass of the fluid. However, it may be calculated using the fluid's density and height. The equation must now be put up. Therefore, the equation for with this pressure is independent of the volume or mass of the liquid. Furthermore, fluid pressure is the product of the density of the liquid, the height of the liquid above the object, and gravity. Because gravity and solid densities are constants, the height of the liquid is the greatest variable in the equation. Furthermore, the pressure difference formula or equation is Pfluid = gh, where denotes the density of the liquid, g denotes the acceleration of gravity, and h denotes the liquid's height (or fluid's depth).

The variables are then multiplied together. To answer the problem, multiply the three variables by their products. A calculator may be used for this purpose.

Consider the instance of a fluid with just a density of (1.08 x 103 kg/m3) and a height of 5.00 m. Now multiply (1.08 x 103 kg/m3) x 9.81 m/s2 (Earth's gravitational acceleration) x 5.00 m. Finally, the solution would be 5.30 x 104.

The findings must now be analysed. Furthermore, one must ensure that the outcomes are logically consistent. Furthermore, there must be no readings for negative fluid pressure.

Different metrics must also be compared to see whether they match the predicted patterns. Higher dense liquids, for example, will exert greater pressure at the same height. Furthermore, as the height is increased, the same liquid will impose additional pressure.

Water is unquestionably denser than oil. As a result, water is expected to exert higher fluid pressure than oil at the very same height.

Fluid Pressure Formula

Pfluid = P plus gh

where,

P denotes the pressure at the reference location.

Pfluid is the pressure at a certain place in a fluid.

is the density of the fluid

g is the acceleration generated by gravity (on Earth, g = 9.8 m/s).

h is the distance from the reference point.

The density of a fluid may be calculated by dividing the fluid's mass by the volume of fluid taken into account.

$$\rho = mv$$

Where,

m is the mass of the fluid.

v denotes the volume of the fluid under consideration.

If the fluid is at atmospheric pressure, the overall pressure on the system is as follows:

$$Po + gh = Pfluid$$

Where

Po denotes atmospheric pressure.

Pressure is a scalar number defined as force per unit area with the force acting perpendicular to the surface. Pressure is a vital physical quantity, with applications spanning from thermodynamics to fluid and solid mechanics. Pressure may be represented in a variety of ways depending on the circumstance.

Fluid pressure is defined as the force per unit area applied to a specific item on the surface of a covered vessel or in the fluid. This pressure is caused by gravity, acceleration, or forces outside of the confined container.

Fluid Pressure Formula

The following relation can be used to calculate the pressure in fluids.

$$P_{\text{fluid}} = P + \rho gh$$

Where,

P = Pressure at the reference point

P_{fluid} = Pressure at a point taken in fluid

P = Density of the fluid

g = Acceleration due to gravity (considering earth g = 9.8 m/s)

h = Height from the reference point

On dividing the mass of the fluid in consideration with the volume of fluid considered, the density of the fluid can be calculated:

$\rho = m/v$

Where,

m = mass of the fluid

v = volume of fluid considered

The total pressure on the system is given as follow if the fluid is subjected to atmospheric pressure:

$P_{\text{fluid}} = P_o + \rho gh$

Where,

P_o = the atmospheric pressure

The Pressure at any Point in a Static Fluid

The total of active forces inside a static fluid at a particular place in space must be equal to zero. Otherwise, the static equilibrium requirement would not be fulfilled. For evaluating such a basic system, consider a rectangular area inside the fluid medium with density L (the same as the fluid medium), breadth w, length l, and height h. The forces operating in this area are then considered inside the medium. To begin, a downward force of gravity (its weight) operating in the area is equal to its density object, times its shape of the object (v), times the acceleration due to gravity (g). Because of the fluid above region, the down force exerted on it is equal to the pressure multiplied by the area of contact. Similarly, owing to the fluid underneath the region, an upward force equal to the pressure times the area of contact acts on this region. To attain static equilibrium, the total of these forces must be zero. To attain static equilibrium, the tension from the fluid below the area must be higher than the pressure from the liquid above by the weight of the region for any region inside a fluid.

Pascal's Principle

Pascal's Principle (also known as Pascal's Law) is applied to static fluids and takes use of the height dependence of pressure in static fluids. Pascal's Principle may be used to hydraulic presses by using the pressure of a static solution as a measure of energy required per volume to do a specific job. Pascal's Principle asserts that pressure is transferred unchanged in a confined static liquid. Pascal's Law may be calculated quantitatively inside a fluid using the formula that calculates pressure at a particular height (or depth) and is specified by Pascal's Principle:

$$p2=p1+\Delta p$$

$$\Delta p=\rho g\Delta h$$

Where,

p1 = pressure applied externally

h = difference in static liquid height

g = gravitational acceleration

Pressure is also in charge of the breathing process and is vital to the respiratory system. Inhalation occurs as a consequence of differences in pressure between the lungs and the environment, which allow air to enter the lungs. The process that causes inhalation is caused by the diaphragm being lowered, which increases the volume of the thoracic cavity enclosing the lungs, reducing its pressure as indicated by the ideal gas law. The decrease in thoracic cavity pressure, which ordinarily has a negative gauge pressure and hence keeps the lungs inflated, draws air into the lungs, inflating capillary alveoli and resulting in oxygen transport required for breathing. The pressure inside the thoracic cavity rises as the diaphragm recovers and goes higher, leading in exhalation. The cycle is repeated, resulting in respiration, which is mechanically caused by pressure changes, as previously stated. Without pressure in the body and the related potential for dynamic biological processes, essential activities such as blood flow and breathing would not be feasible.

Fluid pressure is defined as the pressure seen at a point in a fluid caused by the fluid's weight. It happens in two ways. To begin, there is a natural circulation flow or an open situation. Second, it happened in a closed state or flow. The fluid pressure is also known as static pressure difference or hydrostatic pressure, and it takes the depth of the fluid into account. When considering the fluid's movement, the pressure is insignificant. This indicates that static fluid is unaffected by surface area, container form, or liquid quantity or volume. It should be noted that the term "fluid" refers to a substance's capacity to flow and may refer to both fluids and gases.

Pressure measurement

Pressure measurement is the measurement of a force exerted to a surface by a fluid (liquid or gas). Pressure is commonly expressed as force per unit of surface area. Many ways for measuring pressure and vacuum have been developed. Pressure gauges, vacuum gauges, and compound gauges are mechanical tools used to measure and show pressure (vacuum & pressure). The most common sort of gauge is the Bourdon gauge, which is a mechanical device that measures and signals. A vacuum gauge measures pressures that are less than the ambient air pressure, which is set as the 0, in negative numbers (for example, 1 bar or 760 mmHg equals complete vacuum). Most gauges monitor pressure relative to atmospheric pressure as zero point, hence this kind of measurement is simply referred to as "gauge pressure". Anything more than a complete vacuum, on the other hand, is theoretically a sort of pressure. A gauge that employs complete vacuum as the zero point benchmark must be used at extremely low pressures, delivering pressure readings as absolute pressures.

Gauge Pressure

Gauge pressure, denoted by a 'g' following the pressure unit, e.g. 30 psig, is the most often used pressure reference. Gauge pressure is calculated in relation to atmospheric pressure. Changes in air pressure caused by weather or altitude have a direct impact on the output of a gauge pressure sensor. Positive pressure is defined as a gauge pressure greater than the ambient pressure. When the measured pressure is lower than atmospheric pressure, it is referred to as negative or vacuum gauge pressure. Gauge pressure sensors typically feature a single pressure port. The ambient air pressure is delivered to the rear of the sensor element through a vent hole or a vent tube. A vented gauge pressure transmitter exposes the outside air pressure to the negative side of the pressure detecting diaphragm, ensuring that measurements are always made with reference to the ambient barometric pressure. As a result, when the process level connection is exposed to ambient air, a vented gauge pressure sensor reports 0 pressure.

A sealed gauge reference is nearly similar, except that air pressure is sealed on the diaphragm's negative side.

This is often used in high-pressure applications, such as detecting hydraulic pressures, when variations in ambient pressure have only a little impact on sensor accuracy. The pressure measured via a sealed instrument with the zero point set is known as sealed-gauge pressure. This set point is determined by the manufacturer of something like the sealed pressure gauge based on the pressure within the device prior to sealing.

Absolute pressure

Absolute pressure is the pressure measured in relation to absolute zero pressure, which is the pressure in a complete vacuum. The ideal gas law requires absolute pressure measurement in the same way as temperature must be expressed by its absolute unit, the Kelvin. Many pressure measurement on Earth, such as tyre pressure, remove atmospheric pressure, resulting in gauge pressure. This might be confusing since the pressure is measured in the same units. For observations on Earth, absolute pressure is supplied by the gauge pressure minus atmospheric pressure.

The Formula for Absolute Pressure

Absolute pressure is defined as any pressure found above the absolute zero of pressure. We can measure it using a barometer, and it equals the sum of measurement pressure and ambient pressure.

Thus,

The absolute pressure formula (Pas) is as follows

$$P_a bs = P_a tm + P_g auge$$

Manometer Measure

A manometer gauge is a very simple but highly efficient instrument for measuring pressure. In most circumstances, this will refer to a gauge, which is made out of a U-shaped glass tube filled with mercury or another liquid. Traditionally, one end of the manometer tube is left uncovered, vulnerable to pressure and temperature, while a manometer hose is linked to an extra pressure source through a gas tight seal. A manometer gauge, although often associated with gas pressures, may also be used to gauge the pressure produced by liquids. Because there are no mechanical components in the manometer pressure gauge, it needs no maintenance and is exceptionally precise.

A manometer gas or liquid pressure gauge works on a very basic concept. When a liquid is at rest, the pressure is equal at all points, according to hydrostatic equilibrium. If both ends of the U-tube are left exposed to the environment, the pressure on each side will be equal. As a result, the level of the liquids on the left will be identical to the amount of liquid on the right - balance. However, if one end of the U-tube is left exposed to the environment while the other is attached to an extra gas/liquid source, pressures will vary. A manometer is a device that measures pressure at a specific location in a fluid by balancing the column of fluid with the same or another fluid. It is generally referred to as a U-shaped tube that would be filled with a liquid, gas, steam, or other substance.

You may also use it in a scientific experiment to show the air pressure on a liquid column. This device detects air pressure by employing a container with "U"-shaped letter tubes that are open at either one or both ends. Figure 3.4 Manometer Measure.

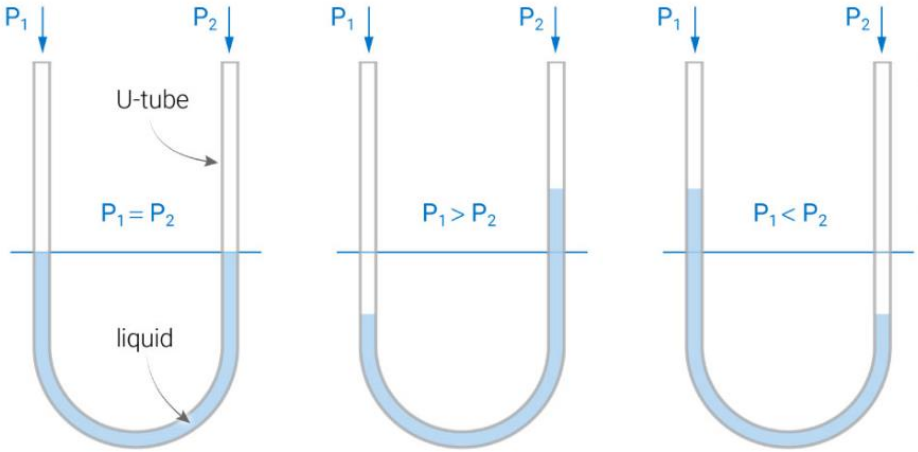

Figure 3.4 Manometer Measure.

If the pressure from the new gas/liquid supply is higher than the ambient pressure, the measuring liquid will be pushed lower. As a result, the liquid will be forced down to the end with higher pressure, forcing it to rise on the other side with lower pressure. If the increased gas/liquid supply provides less stress than the atmospheric pressure, the reverse would occur. In this case, the liquid would fall on the open side of the U-tube and rise on the side linked to the extra gas/liquid supply.

A manometer is a device that measures pressure at a point in a fluid by balancing the column of fluid with the same or another fluid. It is generally referred to as a U-shaped tube that is filled with a liquid, gas, steam, etc. You may also use it as part of a scientific experiment to show the air flow on a liquid column. This is a device that detects air pressure by employing a container with "U"-shaped writing tubes open at either of the ends.

Types of Manometer

Piezometer

To measure the pressure within a vessel or pipe containing liquid, a tube is connected to the container's exterior walls or pipe, allowing liquid to rise in the tube. A pressure transducer of the liquid may be obtained by calculating the height to which the liquid rises and utilising the relation p = gh. Figure 3.5 Piezometer.

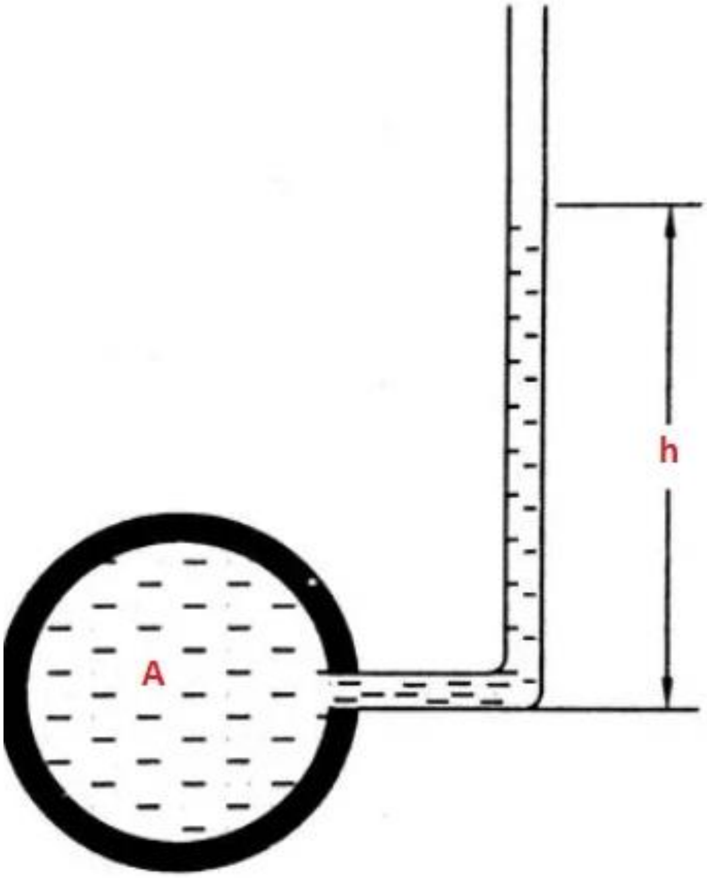

Figure 3.5 Piezometer.

A Piezometer is one such gadget. A piezometer tube must be at least 1/2 inch in diameter to prevent capillary forces. The opening of the instrument must be tangential to every fluid motion; otherwise, an erroneous reading will ensue.

U-tube Manometer

It is made out of a glass tube bent in a V form, with one end linked to a spot where pressure is to be measured and the remaining end exposed to the atmosphere, as shown in the image. Figure 3.6 U-tube Manometer.

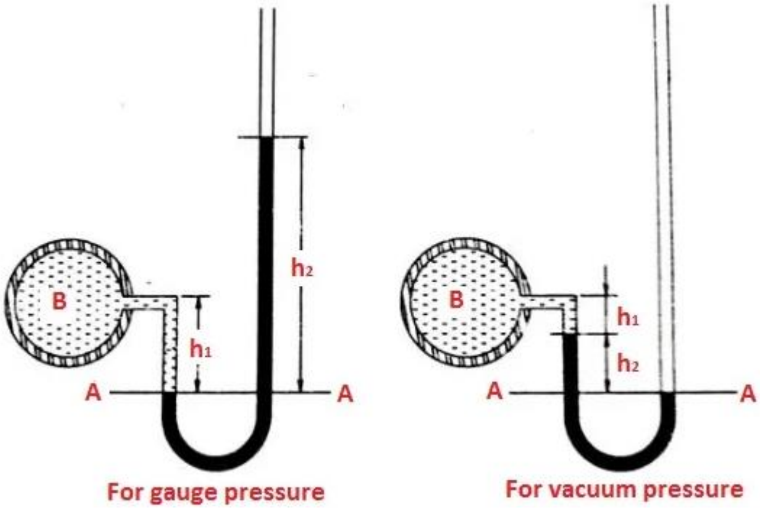

Figure 3.6 U-tube Manometer.

The tube contains mercury and any other liquid or fluid with a substantially greater specific gravity than the liquid whose pressure is being measured.

U-Tube Differential Pressure Manometers

Manometers are pressure measuring devices that use liquid columns in vertical or inclined tubes to measure pressure. One of the most frequent is the water-filled u-tube manometer, which is used to detect pressure differences in pitot or orifices of an air handling or ventilation system's airflow.

The graphic below depicts the water levels in a u-tube when the left tube is linked to a place with greater pressure than the right tube - for example, while the right tube is exposed to the ambient air, the left channel may be connected to a pressurised air duct. Figure 3.7 U-Tube Differential Pressure Manometers

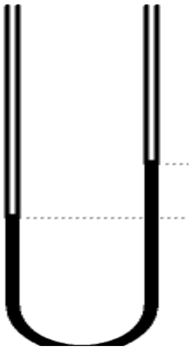

Figure 3.7 U-Tube Differential Pressure Manometers

Vertical U-Tube Manometer

The pressure difference measured by a vertical U-Tube manometer can be calculated as

$$p_d = \gamma h$$

$$= \rho g h \qquad (1)$$

Where

pd = pressure (Pa, N/m2, lb/ft2)

γ = ρ g = specific weight of liquid in the tube (kN/m3, lb/ft3)

ρ = U-tube liquid density (kg/m3, lb/ft3)

g = acceleration of gravity (9.81 m/s2, 32.174 ft/s2)

h = liquid height (m fluid column, ft fluid column)

The specific weight of water, which is the most commonly used fluid in u-tube manometers, is 9.81 kN/m3 or 62.4 lb/ft3.

Inclined U-Tube Manometer

Low column heights and accuracy are typical issues when monitoring pressure differences in low velocity systems or systems of low density fluids, such as air ventilation systems. The accuracy of the u-tube manometer may be increased by inclining it. The illustration below depicts a u-tube with the left tube linked to a greater pressure than the right tube. It is important to note that the left and right tubes must be in the same declining plane in order for the angle towards the horizontal plane to be proper (Figure 3.8).

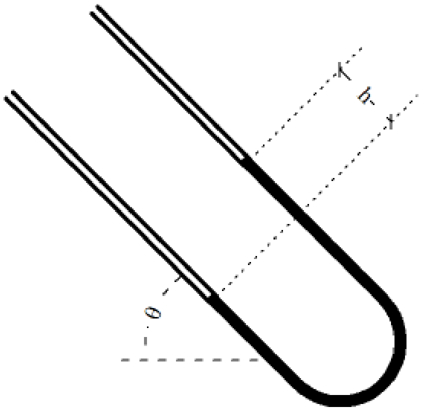

Figure 3.8 Inclined U-Tube Manometer

The pressure difference in an inclined u-tube manometer can be expressed as

$$pd = \gamma\ h \sin(\theta) \qquad (2)$$

Where

h = length, difference in position of the liquid column along the tube (mm, ft)

θ = angle of column relative the horizontal plane (degrees)

Inclining the tube manometer increases the accuracy of the measurement.

Single Column Manometer

Consider a vertical tube micromanometer attached to a high-pressure pipe holding a light liquid. Because of the pressure in the pipe, the lighter liquid in the basin will push the heavier liquid downwards. Figure 3.9 Single Column Manometer.

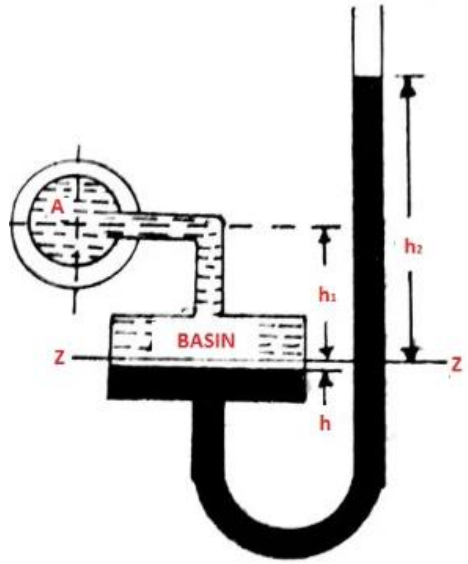

Figure 3.9 Single Column Manometer.

The decline of a heavy liquid surface will be quite little due to the bigger size of the basin. This transfer of heavy liquid into the basin will cause a large increase in heavy liquid in the right limb.

Inclined Tube Manometer

An inclined tube micromanometer is one that has the vertical tube of the micromanometer tilted as illustrated in the illustration (Figure 3.1).

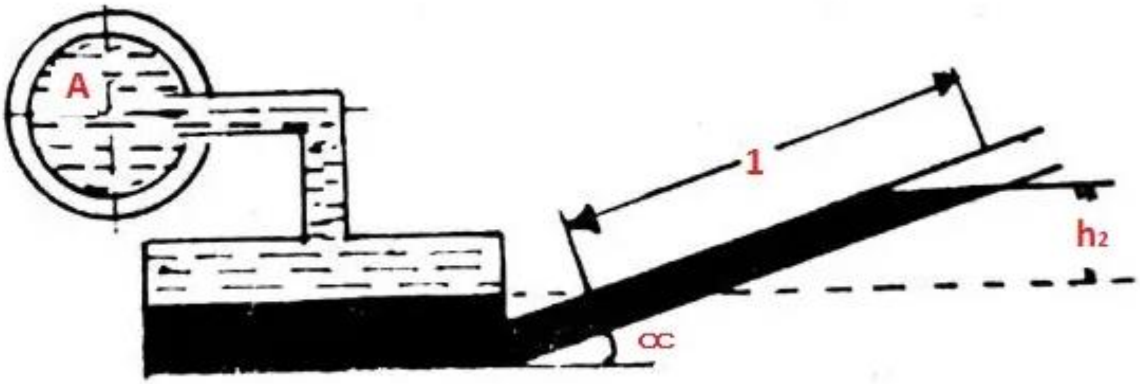

Figure 3.10 Inclined Tube Manometer.

The vertical tube kind of inclined micromanometer is less sensitive. Because of the slant, the heavier liquid moves a greater distance in the right limb. As a result, it might provide a greater reading for such given pressure.

Differential Manometer

A differential manometer is a device that measures the tension difference between two places in the same or distinct pipes. A differential manometer is made out of a U-tube carrying a heavy liquid and two ends joined by pressure difference measuring points:

It comprises of two piezometers installed at two distinct gauge sites to measure the pressure differential. The difference in liquid levels of the two tubes may be used to calculate the pressure differential between the two places. It has certain constraints in the kind of piezometers.

Differential Manometer with U-tube

It is a device that measures the difference in pressure between two places in a pipe or between two distinct pipes. This manometer is made out of a U-shaped tube filled with a heavy liquid.

Manometer for differential pressure in a U-tube

The two ends are linked to the two spots in the pipe where a pressure differential is necessary. Allow pressure at point A to be greater than pressure at point B. The higher pressure at A will then cause the heavier liquid in the U-tube to travel downward. The heavy water in the left limb will travel lower, causing the heavy liquid in the right limb to ascend. Figure 3.11 Manometer for differential pressure in a U-tube.

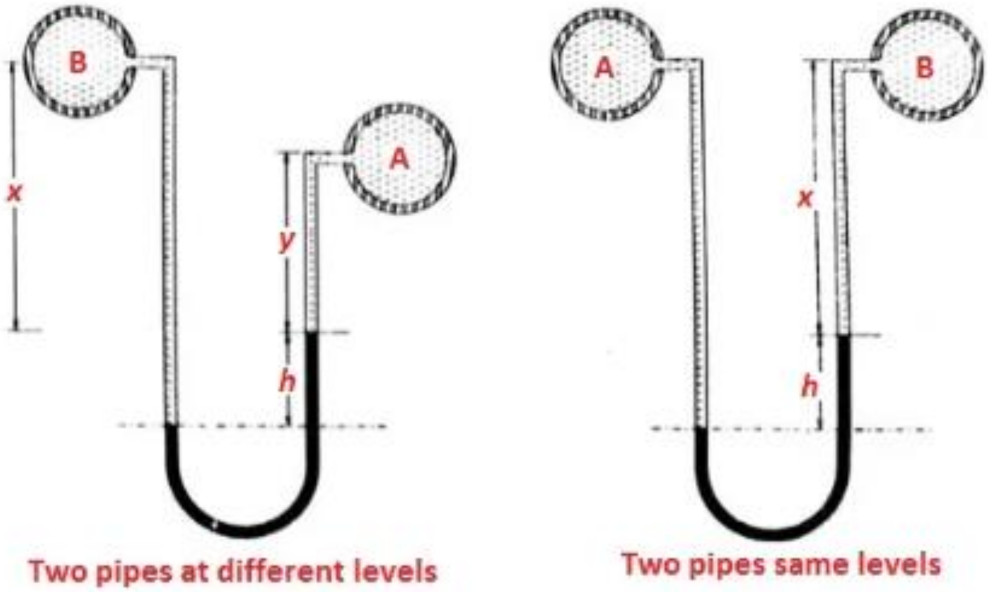

Figure 3.11 Manometer for differential pressure in a U-tube

Manometer with Inverted Differential

The U-tube of these manometers is inverted and filled with a light liquid. The tube's two ends are linked to the spots where the pressure differential is to be monitored.

Differential manometers with inverted U-tubes

It is used to determine the difference between low pressures. The diagram depicts an inverted U-tube with a differential manometer attached to points A and B. Assume that the pressure at point A exceeds the pressure at point B. Figure 3.11 Differential manometers with inverted U-tubes.

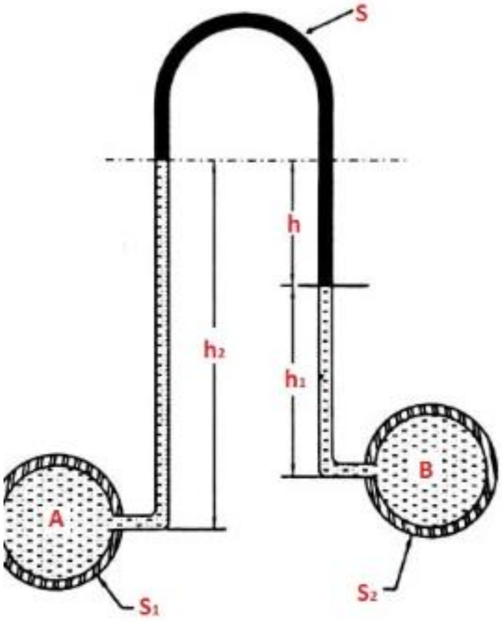

Figure 3.11 Differential manometers with inverted U-tubes.

Small Manometer

These are manometers that operate on the basis of an inclined tube manometer. Little manometers or gauges are used to measure extremely small pressure or pressure fluctuations. These voltmeter are also known as micro-manometers, which are a modified variety of a basic manometer with a huge cross-sectional space as its portion. It is a very accurate device capable of detecting extremely minute pressure fluctuations.

Advantages of Manometers

Following are the main advantages of manometer:

1. It is simple to construct.
2. It has great accuracy.

CHAPTER 4

FLUID PROPERTIES

Mr. Raminder Singh,
Associate Professor, Department of Chemical Engineering, Jaipur National University, Jaipur India
Email Id- singh9raminder@jnujaipur.ac.in

A material that constantly deforms when subjected to external force is fluid. These are the materials that are unable to withstand the shear force (a force that causes a change in form) that is applied to them. Toothpaste, molten lava, air, water, etc. Under the influence of force or pressure, a fluid flows.

First, we must clarify what is involved in a process "fluids" in order to comprehend the many qualities of fluids. Everything that can stream is a fluid, by definition. Fluids include things like the water we drink or the air we breathe. Fluids are essentially all gases and liquids. Let's gain in-depth information about the characteristics of fluids in this post.

Kinematic Properties

A branch of physics called kinematics, which was evolved from classical mechanics, defines how points, bodies, and systems of organisms (groups of entities) move without taking into account the forces that propel them. The fluids' kinematic characteristics are speed and acceleration. Thermodynamic characteristics: These characteristics aid in comprehending the fluid's thermodynamic condition. Thermodynamic characteristics of fluids include temperature, density, pressures, and specific enthalpy

Thermodynamics Properties

These characteristics aid in comprehending the fluid's thermodynamic condition. The thermodynamic characteristics of fluids include their temperature, density, pressure, and specific enthalpy. Physical characteristics: These characteristics, including as colour and odor, aid in recognizing the fluid's physical state.

Physical properties

There are three physical properties of fluids that are particularly important: density, viscosity, and surface tension. Each of these will be defined and viewed briefly in terms of molecular concepts, and their dimensions will be examined in terms of mass, length, and time. These characteristics, such as coloration and odor, aid in understanding the fluid's physical state.

Density

The mass per unit volume of a substance or item is its average density.

$$\rho=M/V$$

Specific weight

The weight of a fluid per unit volume is known as specific weight, often referred to as unit weight. It has dimensions in terms of pressure is defined volume and is typically represented by that of the Greek letter (gamma).

Temperature

One of the thermodynamic characteristics of fluids what determines whether they are in a hot or cold state is their temperature. Fahrenheit, Celsius, or Kelvin are all used to measure temperature.

Pressure

The pressure in fluids may be determined utilizing relation: $P = h\rho g$ (Pressure = Height or Depth of something like the liquid × Density of something like the liquid × Centrifugal pull (9.81m/s)). A scalar quantity, pressure. The Pascal, often known as the Newton per square metre (N/m2), is the SI (International System of Units) unit of pressure. A fluid's pressure is defined as the force it exerts.

$$F/A$$

Specific Volume

Specific volume is the reverse of density in fluid mechanics. It is the amount of space a fluid takes up per unit of mass. The reciprocal of the material's density, or the mass by unit volume, is the specific volume, or $r = (1/v) = (m/V)$. A substance's "Specific Gravity" is determined by dividing its mass by the mass of an equivalent flow of liquid at the same pressure and temperature.

Surface tension

The forces of attraction between liquid particles are what create surface tension. Although the attraction between the atoms in a drop of water is minimal, the attraction between the molecules inside the drop is equal. Other water molecules inside the drop are drawn to a surface molecule. Surface tension is the phenomena that occurs when a liquid's surface comes into contact with yet another phase. The temperature of water at 100°C is 0.059 N.m-1 while at 0°C it is 0.079 N.m-1. The tensile force exerted on the liquid surface in contact with just a gas or other substance is referred to as surface tension. In such a way that the contact contact between two or more substances liquids functions as a membrane under strain. The magnitude of this produced per unit length of something like the free surface have the same value as that of the surface energy per unit area. The Greek letter is used to represent it (called sigma)

A solid's molecules are often closer together than a fluid's. The considering the size of the attraction forces between a solid's molecules, a solid tends to to keep its form. This is not true for fluids since they have less molecular attractive forces. Under load, a perfect elastic solid will deform. And will revert to their initial state when the burden has been eliminated. Some solids are Plastic. These distort when a sufficient load is applied, and the deformation lasts as long as the load is supplied as long as the material doesn't tear. When the stress is released, there is no longer any deformation, yet the plastic solid doesn'real reappear to its initial condition.

This force has little effect when a substance is in the liquid condition because molecules are drawn in all directions to one another. However, because there aren't any molecules to draw the others up, none of the molecules are being dragged up at the liquid's surface. The molecules are being pulled down by a powerful force that exists between them. As a result, a robust barrier is formed at the liquid-gas contact. A liquid's surface must be broken through with far more force than the liquid itself.

Fluid Statics

When the fluid velocity is zero, known as the hydrostatic state, the pressure change is caused only by the fluid's weight.Consider a tiny wedge flowing fluid at rest with dimensions x, z, s, and b into the paper. By definition, there is no shear stress, and pressure is supposed to be the same on each face (small element), as shown in Figure 4.1.

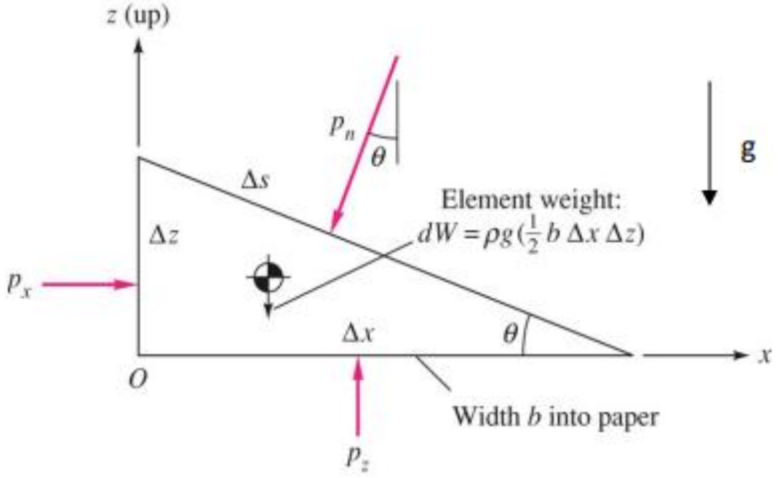

Figure 4.1 Fluid Statics.

Because the atom is at rest, the total of all forces must be 0.

$$\sum F_x = 0 = p_x\, b\, \Delta z - p_n b\, \Delta s\, sin\theta$$

$$\sum F_z = 0 = p_z\, b\, \Delta x - p_n b\, \Delta s\, cos\theta - \frac{1}{2}\rho g\, b\, \Delta x\, \Delta z$$

Derived from geometry, Substitution in the above equations yields:

$$p_x = p_n \qquad p_z = p_n + \frac{1}{2}\rho g\, \Delta z$$

Fluid Kinematics

Fluid flow kinematics is concerned with the flow of fluid particles without regard for the agent causing the motion. This is about the geometry of fluid particle motion. This also addresses the velocity and acceleration of moving fluid particles. The motion of a fluid may be analysed using the same ideas as the mobility of a solid.

However, there is a fundamental distinction between the movement of a solid and the motion of a fluid. A workpiece is dense and moves as a single mass. A solid body's particles do not move relative to one another. As a result, we investigate the motion of the complete body rather than the motion of each atom of a solid body. However, in the case of a fluid body, the fluid particles are all individually mobile and have their own movements. A fluid particle's velocity may vary from that of its surroundings. However, a link between the movements of nearby fluid particles may be feasible.

Methods of Describing Fluid Motion:

Each particle of a fluid in motion has a specific value of its attributes like as density, velocity, acceleration, and so on at any given time. As the fluid travels, the values of these attributes will shift from one place to another at random.

As a result, it is feasible to recognise two ways for describing fluid motion. The first approach, known as the Lagrangian method, investigates the velocity, acceleration, and other properties of a single fluid particle at each instant of time as it travels to various places. Because analysing the characteristics of a single real fluid is a time-consuming operation, this approach is not widely used. The second way, known as the Eulerian method, describes flow by investigating velocity, acceleration, pressure, density, and so on at a given location in space. This approach is the most often used since it is simple to implement.

Let x, y, and z represent spatial coordinates, and t represent time. Let V be the resulting velocity in a fluid body at any location in space. Let u, v, and w represent the components of the resulting velocity V at any location along the x, y, and z axes. The notations are shown in the figure below. Figure 4.2 Describing Fluid Motion

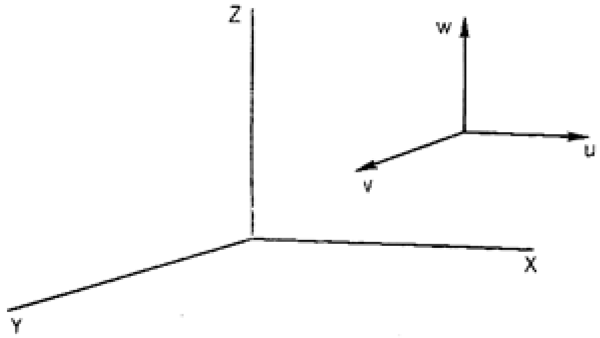

Figure 4.2 Describing Fluid Motion.

Stream Line:

A stream line is a continuous line in a fluid that indicates the direction of the fluid's velocity at each location along the line. At every location along the stream line, the tangent is in the direction of the velocity at that point. Fluid particles on a stream line travel along the stream line at the same time. At a point, the velocity vector's direction does not change. In other words, the location of the stream line is set. In contrast, if the stream line pattern is constant, the flow is constant. In the case of an unstable flow, the velocity direction varies with time at each site. This indicates that the location of a stream tube does not remain constant. A stream line's position varies from instant to instant.

Path Line

A path line is the route or line that is really defined by a single flowing fluid as it travels through time. The route line represents the velocity of an identical fluid particle at different points in time. Because there are no changes in velocity in a constant flow, the route line corresponds with the stream line. In the event of an uneven flow, the stream lines change locations at every moment, causing the route line to oscillate between multiple stream lines over time.

Streak Line:

The streak line is the location of fluid particle locations that have passed through a specific place in succession. Assume A, B, C, D... are fluid particles that successively passed past a reference point, say the origin. These particles have drawn their own paths. Assume that such particles A, B, C, D... are at Pa at time t, as shown in Figure 3.3.

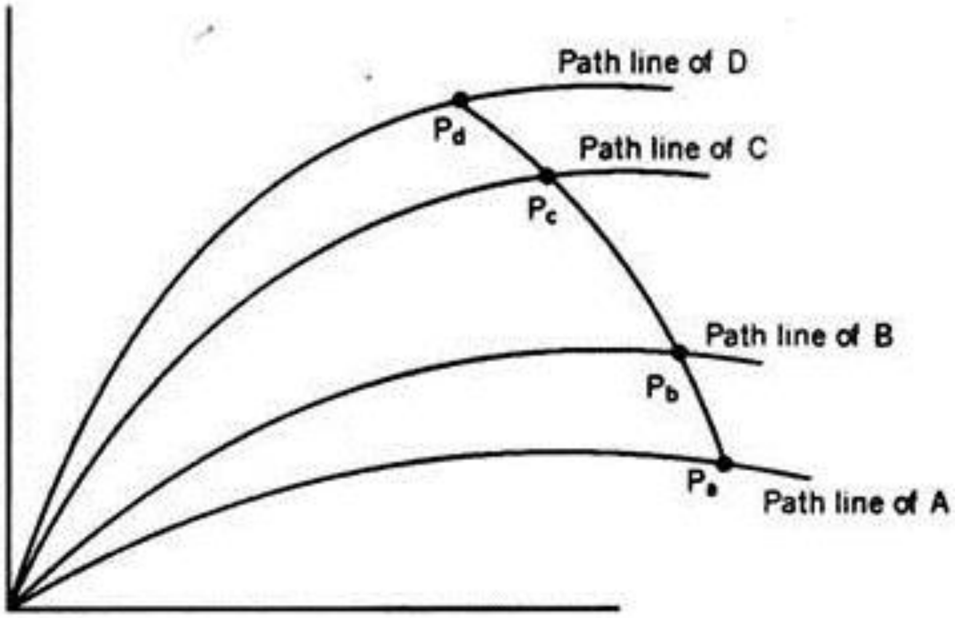

Figure 3.3 Streak Lines.

Stream Tube

A stream tube is a tubular space produced by a series of stream lines. It is a grouping or collection overlapping stream lines that create a tabular area. Stream lines make form the surface of either a stream tube. A fluid particle's velocity on the surface of a stream tube is along the stream line across the surface of the stream tube. This indicates that no flow may occur over the walls of such a stream tube. The velocity at the centroid of any portion of the stream tube indicates the average velocity of flow through the flow stream at that region. A stream tube is seen in the figure below. Because there can be no flow over the walls of such a stream tube, it should be obvious that the amount of fluid entering a stream tube equals the amount of fluid exiting the stream tube. Figure 3.4 Stream Tube

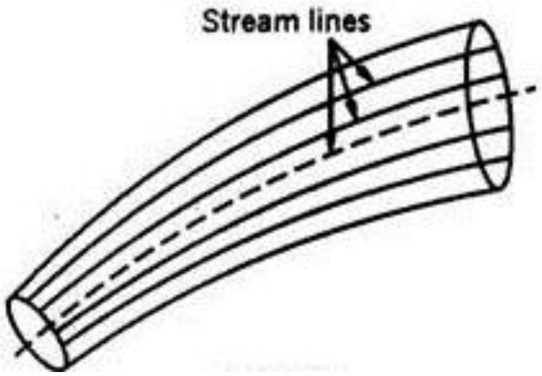

Figure 3.4 Stream Tube.

Laminar Flow

Laminar flow, also known as streamline flow, is a form of fluid (gas or liquid) movement in which the fluid moves smoothly or in regular patterns, as opposed to turbulent flow, in which the fluid fluctuates and mixes irregularly. The velocity, pressure, and certain other flow parameters at each location in the fluid stay constant in laminar flow. Laminar flow across a level surface may be conceived of as a series of parallel thin layers, or laminae. The fluid in interface with the horizontal surface remains fixed, whereas the other layers glide over one another. As one example, a fresh deck of cards may be designed to "flow" laminarly.

Laminar flow in a straight pipe may be thought of as the relative motion of a series of concentric cylinders of fluid, the outer one stationary to the pipe wall and the others flowing at increasing rates as the pipe's centre approaches. Laminar flow occurs when smoke rises in a straight direction from a cigarette. After a short distance, the smoke normally turns to a turbulent flow as it planipennis and swirls away from its normal route. Figure 3.5 Laminar Flow

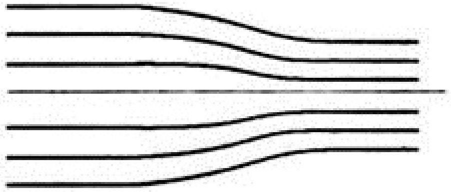

Figure 3.5 Laminar Flow

Turbulent flow

This is the most prevalent kind of flow seen in nature. This flow is distinguished by the chaotic, irregular, and unexpected motion of fluid particles, which produces eddy currents. In motion, there is a general conflating of fluid particles. The velocity varies in both direction and intensity from one place to the next. Particles are constantly colliding, causing momentum to be transferred between them. When compared to the energy loss in laminar flow, eddy currents cause a significant loss of energy. This increased energy loss is due to turbulent shear stresses being substantially larger than the laminar shear stresses described by Newton's equation of viscosity. Figure 3.6 Turbulent flow

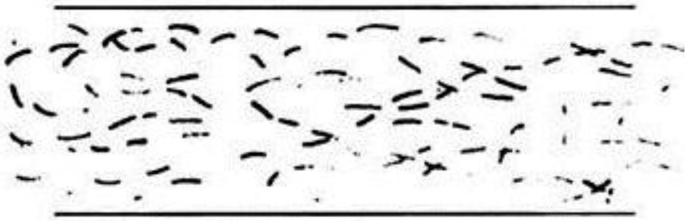

Figure 3.6 Turbulent flow.

Turbulence is identified in an eddy currents by its irregularity, uncertain frequency, and lack of a distinct discernible pattern. This sort of flow cannot be completely mathematically analysed, however statistical analysis is conceivable. River water flow is often tumultuous. High-velocity water flow in pipes is turbulent. The flow of viscous oil in narrow tubes, ground water flow, and blood flow in blood arteries are all laminar. As the water's velocity in a pipe increases gradually, the flow transitions from laminar to turbulent. The critical velocity in a pipe is the rate at which the flow transitions from laminar to turbulent. The kind of flow that occurs in any scenario is determined by the value of a non-dimensional number dv/ known as the Reynolds' number, where d is the size

Continuity principle

The continuity principle, often known as the continuity equation, the fluid mechanics principle. Simply put, what comes into a specified volume in a defined period must accumulate in that volume, less what flows into that volume in that time. If the accumulation has a negative sign, the substance in that amount is being depleted. The principle is a result of the law of mass conservation. This equation, together with a second equation based on the second of Newton's principles of motion and a third equation based on energy conservation, completely refers to the behavior of fluids in motion.

Turbulence

A flow situation in fluid mechanics in which local speed and pressure alter unexpectedly although an average flow is maintained. Wind and water spinning around objects are common examples, as is any kind of rapid flow (Reynolds number more than 2,100). Turbulence has properties such as eddies, funnels, and a decrease in drag. Golf balls with lower drag may fly further than they would otherwise, and their dimpled surface is designed to create turbulence in the boundary layer. If, as stated, swimsuits featuring rough surfaces assist swimmers go quicker, the same rationale may apply. The flow is normally laminar when the Reynolds ratio is less than 2000. When the Mach number exceeds 2800, the flow is considered turbulent. If the Reynolds number is between the upper and lower bounds, the flow might be laminar or turbulent. As a result, there is no set or defined value for the critical velocity. The velocity relating to a Reynolds number of 2000 is referred to as the lower critical velocity, while the velocity corresponding to a Reynolds number of 2800 is referred to as the upper critical velocity.

Steady Flow and Unsteady Flow

Steady Flow:

The flow is considered to be steady if the flow properties like as velocity, density, pressure, and so on at a particular place in a flowing mass of a fluid do not vary with time. On the other hand, if these flow parameters at a particular site fluctuate over time, the flow is said to be unstable.

Because velocity is a widely used flow characteristic, it is sufficient to consider the flow steady if a velocity at a particular place does not vary with respect to time. Assume V is the velocity at a given place (x1, y1, z1). The flow is steady at this moment if V stays constant at all times. However, if V varies with time at this point, the flow is unstable.

Uniform and Non-Uniform Flow

The flow is uniform if the flow properties, such as velocity, density, and pressure, stay constant at all places at a given moment. If V is selected as a flow characteristic, V possesses the same value at all places and is independent of spatial position at any given moment. Non-uniform flow occurs when the flow characteristics have varied values at various moments in time.

In the context of open channels, we often use the words uniform and non-uniform flow. The flow will be uniform in a channel when the section of the channel is uniform and the depth of flow is uniform because the velocity will be the same at all sections. However, if the sectional widths of the channel alter at various sections, the flow depths will differ as well. Obviously, the velocity will vary across portions, and the flow will be non-uniform, whether the flow is uniform or non-uniform. If the flow rate is constant, the flow is steady; if the flow rate fluctuates over term, the flow is unstable. As a result, we may encounter steady or uneven flow, uniform or non-uniform flow. Any form of flow may exist in isolation from the others. It is also feasible to combine two forms of flow.

Rotational and Irrotational Flows:

Fluid particles may be susceptible to translatory or rotatory dispersion while the fluid flows. If a particle travelling down a stream line spins along its own axis, the particle is said to be in rotational motion. The particle is considered to have unsteady motion if it does not rotate around its own axis as it flows down the stream line (Figure 4.7).

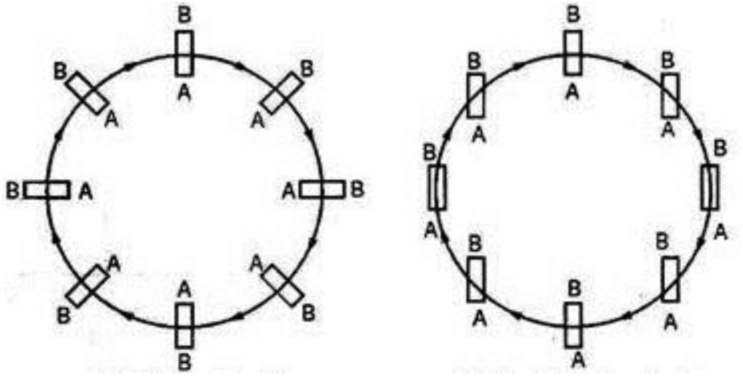

Figure 4.7 Rotational and Irrotational Flows.

Types of Fluid Flow are thorough classifications of numerous fluid flows seen in nature. We define fluid flow based on features such as rolling motion, pressure drop, density fluctuation, pathways, and so on. Fluid Flow is a fluid mechanics field that deals with fluids. It entails fluid movement under the force of uneven pressures. This motion will continue as long as uneven forces are applied. Fluid Mechanics distinguishes six kinds of fluid flow.

Before delving into the complexities of the many forms of flow in fluid mechanics, it's essential to first comprehend fluid. A fluid is a liquid, gas, or other substance that deforms (flows) continually as a result of applied shear stress or external force. They have a zero modulus and strength, which means they can't withstand any shear stress.

Fluid Flow

Fluid flow is a branch of fluid mechanics that deals with fluid dynamics. This phenomenon is due to the movement of a fluid exposed to unbalanced forces. This motion will continue as long as uneven stresses are applied. Let's look more closely at the many forms of fluid flow. Fluids have been classified into five types: ideal fluid, real fluid, Newtonian fluid, non-Newtonian fluid, and optimal plastic fluid. There are two methods for analysing fluids.

The Lagrangian Approach: In this technique, a single fluid particle is chosen and its behaviour is investigated in several domains (with respect to space).

The Eulerian Approach: This technique examines the behaviour of fluid particles passing across a piece of space at various times. This approach is often used in Fluid Mechanics.

One, Two and Three Dimensional Flows

The terms one, two, and three dimensional flow relate to the number of coordinates necessary to describe a flow. Any physical flow looks to be three-dimensional in general. These, however, are difficult to compute and need as much simplification as possible. This is accomplished by disregarding changes in flow in either direction, hence minimising complexity. It is sometimes feasible to convert a three-dimensional issue to a two-dimensional problem, and even to a one-dimensional problem (Figure 4.8).

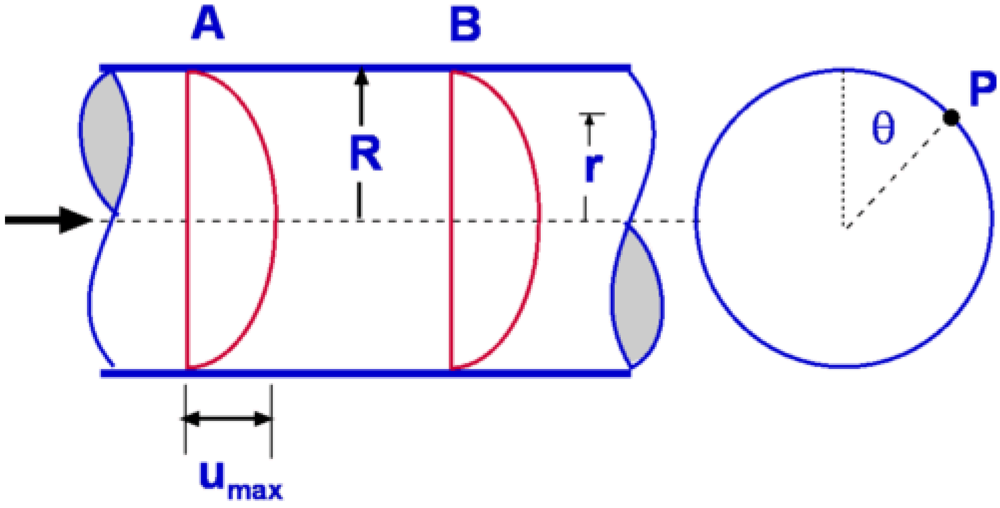

Figure 4.8 One, Two and Three Dimensional Flows.

Consider the flow of water via a circular conduit. This flow is complicated at the point when it enters the pipe. However, as we go downstream, the flow becomes simpler and reaches the condition of a fully evolved flow. One feature of this flow is that the velocity remains invariant with in flow direction, as seen below. This flow's velocity is determined by:

$$u = u_{max}\left[1 - \left(\frac{r}{R}\right)^2\right]$$

It is clear that velocity at any point is determined only by the radial distance from the centerline and is unaffected by distance, x, or angular position. This is an example of a typical one-dimensional flow. Consider the flow in a diverging duct a. The velocity at each point is determined not only by the radius but also by the x-distance. As a result, this is a two-dimensional flow. Figure 4.9 One, Two and Three Dimensional Flows impact.

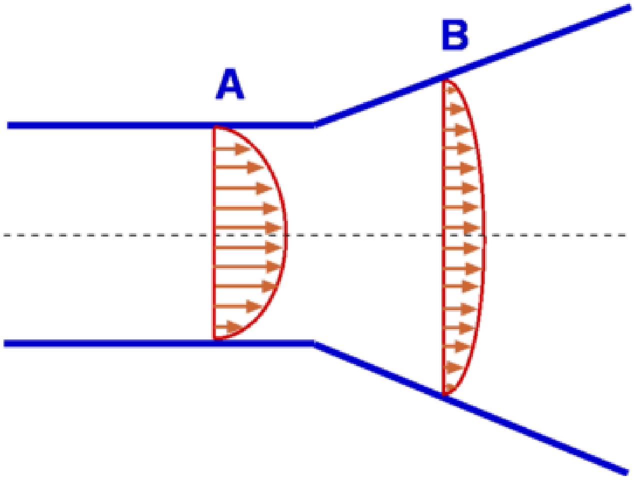

Figure 4.9 One, Two and Three Dimensional Flows impact.

Continuity Equation

The continuity equation describes the transmission of different quantities such as fluid or gas. The formula explains how a flowing fluid conserves mass. The equation describes how a fluid saves energy mass while moving. The continuity equations may be used to illustrate the conservation of many physical phenomena, such as energy, mass, motion, natural integers, and electric charge. The continuity equation provides essential information regarding fluid flow and behaviour as it passes through a pipe or hose. The Governing Equations is used to a wide range of things, such as tubes, pipelines, rivers, and ducts that transport gases or liquids. The continuity equation may be expressed in either differential or integral form, depending on whether it is applied at a point or in a finite area. Developing the Continuity Equation.

Principle of continuity

The transfer of various quantities, such as fluid or gas, is described by the continuity equation. The equation, for example, illustrates how a fluid helps to conserve mass while moving. The continuity equations retain many physical phenomena such as energy, matter, momentum, natural numbers, and electric charge. Figure 4.10 the Equation of Continuity.

Deriving the Equation of Continuity

$$m = \rho_{i1}\, v_{i1}\, A_{i1} + \rho_{i2}\, v_{i2}\, A_{i2} + \ldots\ldots + \rho_{in}\, v_{in}\, A_{im}$$

$$m = \rho_{o1}\, v_{o1}\, A_{o1} + \rho_{o2}\, v_{o2}\, A_{o2} + \ldots\ldots + \rho_{on}\, v_{on}\, A_{om} \ldots\ldots\ldots\ldots (1)$$

Where,

m = Mass flow rate

ρ = Density

v = Speed

A = Area

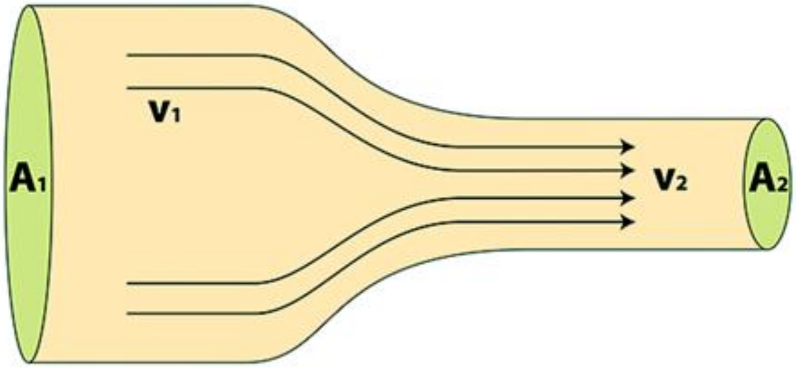

Figure 4.10 the Equation of Continuity.

With uniform density equation (1), it can be modified to:

$$q = v_{i1}\, A_{i1} + v_{i2}\, A_{i2} + \ldots. + v_{in}\, A_{im}$$

$$q = v_{o1}\, A_{o1} + v_{o2}\, A_{o2} + \ldots. + v_{on}\, A_{om} \ldots\ldots\ldots.. (2)$$

Where,

q = Flow rate

Fluid Dynamics

Fluid dynamics is defined as "the field of practical science dealing with the flow of liquids and gases" by the American Heritage Dictionary. It is used to calculate force and moments, determine the mass flow rate of petroleum via pipelines, forecast weather patterns, explain nebulae in intergalactic space, and simulate fission bomb explosion. Fluid dynamics is a subdiscipline of fluid mechanics that explains the movement of fluids liquids and gases in physics, physical chemistry, and engineering. Aerodynamics (the study of air or other gases in motion) and fluid mechanics are two of its subdisciplines (the study of liquids in motion). Fluid dynamics is used to calculate stresses and moments on aeroplanes, determine the mass flow rates of petroleum via pipelines, forecast weather patterns, explain nebulae in interstellar space, and simulate fission bomb explosion.

Fluid dynamics provides a systematic framework that underpins these practical disciplines, including empirical and semi-empirical rules obtained from flow measurement and applied to practical applications. A fluid dynamics issue is often solved by calculating different fluid parameters such as fluid velocity, pressure, density, and warmth as functions of location and time. Prior to the twentieth century, hydrodynamics was interchangeable with fluid dynamics. This may still be seen in the titles of several fluid dynamics subjects, such as mhd flow and hydrodynamic stability, which can both be applied to gases. The conservation rules, especially mass conservation, conservation of linear momentum, and conservation of energy, are the fundamental axioms of fluid dynamics (also known as the First Law of Thermodynamics). These are based on classical physics and have been altered by quantum mechanics and general relativity. The Reynolds transport theorem is used to express them.

In addition to the above assumptions, fluids are expected to follow the continuum assumption. Fluids are made up of molecules colliding with one another as well as solid things. The continuum assumption, on the other hand, posits that fluids are continuous rather than discrete. As a result, parameters like as density, pressure, temperature, and flow velocity are supposed to be well-defined at infinitesimally tiny places in space and to fluctuate continuously from one point to another. The fact that perhaps the fluid is composed of separate molecules is overlooked. The momentum equations for Newtonian fluids are the Navier-Stokes equations, which are a non-linear set of differential equations that describe the flow of a fluid whose stress depends linearly on flow velocity gradients and pressure for fluids that are sufficiently dense to be a continuum, do not contain ionised species, and have flow velocities that are small in relation to the speed of light. Because the unsimplified equations lack a generic closed-form solution, they are mainly useful in computational fluid dynamics. The equations may be reduced in a variety of ways, making them simpler to solve. Some of the simplifications enable the solution of basic fluid dynamics problems in closed form.

To fully characterise the situation, a thermodynamic equation of state that specifies the pressure as a function of other thermodynamic variables is necessary in addition to the mass, momentum, the energy conservation equations. The ideal gas equation of state is an example of this:

$$p = \frac{\rho R_u T}{M}$$

CHAPTER 5

VELOCITY POTENTIAL FUNCTION AND STREAM FUNCTION

Miss Isha Srivastava,
Assistant Professor, Department of Chemical Engineering, Jaipur National University, Jaipur India
Email Id- isha@jnujaipur.ac.in

The velocity potential function and the stream function are two numeric functions that may be used to determine whether a fluid flow is rotational or irrotational. Both functions provide a unique Laplace equation. Depending on whether or not the Laplace equation is satisfied, the fluid flow might be rotational or irrotational.

Velocity Potential Function

The velocity potential function is a scalar space-time function. If the velocity potential function is represented by 'phi,' then the mobility function for just a steady fluid flow is given by the statement.

$$\Phi = f(x, y, z)$$

It is a scalar function whose negative derivative yields the flow velocity in that direction.

$$u = -\frac{\partial \varphi}{\partial x}\,;\; v = -\frac{\partial \varphi}{\partial y}\,;\; w = -\frac{\partial \varphi}{\partial z};$$

The velocity components of the fluid flow along the x, y, and z axes are represented by u, v, and w.

$$u_r = \frac{\partial \varphi}{\partial r}\,;\; u_\theta = (1/r)\frac{\partial \varphi}{\partial \theta}$$

In polar coordinates, the velocity order to be successfully is written as,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0;$$

Denote the velocity vectors in the radial and tangential directions, i.e. along r and 'theta, the following formula yields the 3D Laplace equation in the context of velocity potential function.

$$\frac{\delta^2\varphi}{\delta x^2}+\frac{\delta^2\varphi}{\delta y^2}+\frac{\delta^2\varphi}{\delta z^2}=0$$

The following formula expresses the Laplace equation in the form of the velocity potential function in 2D.

$$\frac{\delta^2\varphi}{\delta x^2}+\frac{\delta^2\varphi}{\delta y^2}=0$$

If the velocity potential function obeys the Laplace equation, it relates to a kind of fluid flow.

Properties of Velocity Potential Function

Understanding the rotatable components along the x, y, and z axes helps to explain the characteristics of a fluid's velocity potential function. It is provided by:

$$\mathrm{w_z}=\frac{1}{2}\left[\frac{\delta v}{\delta x}-\frac{\delta u}{\delta y}\right]$$

$$\mathrm{w_y}=\frac{1}{2}\left[\frac{\delta u}{\delta z}-\frac{\delta w}{\delta x}\right]$$

$$\mathrm{w_x}=\frac{1}{2}\left[\frac{\delta w}{\delta y}-\frac{\delta v}{\delta z}\right]$$

We get the rotational parts by inserting the values of u, v, and w.

$$\mathrm{w_x} = \mathrm{w_y} = \mathrm{w_z} = 0;$$

We assume that the velocity potential function "phi" is a continuous function in this case.

It is clear from the preceding discussion that, the rotational components of the fluid flow must be equal to zero for the flow to be irrotational. If a velocity potential exists, the flow rate is rotational. If the supplied velocity potential obeys the Laplace equation the fluid flow is a model of stable incompressible irrotational flow.

Stream Function

A stream function is a scalar space-time function whose derivative with respect to any direction gives the fluid velocity at right angles to that direction. It is denoted by the symbol "psi," where

$$\Psi = f(x, y)$$

The stream function in cylindrical polar coordinated is given by,

$$u_r = (1/r)\frac{\partial\psi}{\partial\theta}\,;\; u_\theta = -\frac{\partial\psi}{\partial r}$$

Where, u_r and uo radial and tangential velocity.

Lines of Equipotential

The equipotential line is the line along which the velocity potential function is constant. dy/dx = -u/v is the slope of the equipotential line.

Streamlines are the lines along which the stream function remains constant. A tangent drawn at any point on a streamline in a flow field reveals the velocity direction. As a result, dy/dx=v/u gives the slope at any point along the streamline. The above argument is the streamline defining equation. An endless number of streamlines may be drawn with a constant stream function value. The flow patterns may be readily viewed given a family of streamlines. Furthermore, all of the streamlines are opposite to one another. The product of the slope of the equipotential line and the streamline is -1. This signifies that the two lines are orthogonal to one another. As a result, knowing the value of the stream function determines the velocities potential value or vice versa. Figure 5.1 Lines of Equipotential.

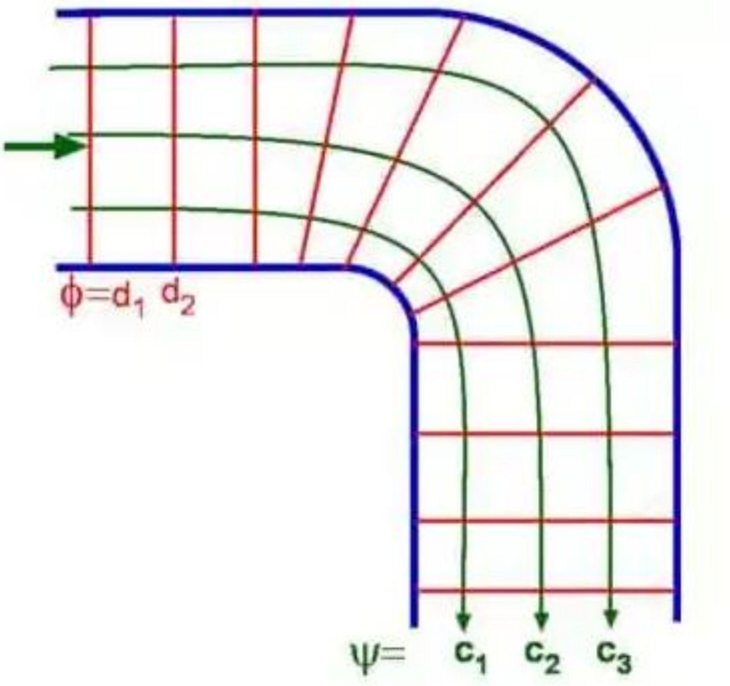

Figure 5.1 Lines of Equipotential.

Conservation of Momentum, Mass, and Energy

There are several mathematical models that explain fluid flow, as well as engineering correlations that may be employed in certain instances. However, partial differential equations provide the most full and accurate explanation (PDEs). A flow field, for example, is defined by mass, momentum, and total energy balance as represented by the formula, the Riemann equations, as well as the total energy equation:

$$\begin{aligned}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0 \\
\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) &= -\nabla p + \nabla \cdot \tau + \mathbf{F} \\
\frac{\partial}{\partial t}\left[\rho\left(e + \frac{1}{2}u^2\right)\right] + \nabla \cdot \left[\rho \mathbf{u}\left(e + \frac{1}{2}u^2\right)\right] &= \nabla \cdot (k \nabla T) + \nabla \cdot (-p\mathbf{u} + \tau \cdot \mathbf{u}) + \mathbf{u} \cdot \mathbf{F} + Q
\end{aligned}$$

The mathematical model equations' solutions provide the velocity field; pressure, p; and temperature, T; of the fluid in the modelled domain.

In theory, this set of equations may represent flows ranging from creeping flow in a microfluidic device to turbulent flow in a heat exchanger and supersonic flow surrounding a jet fighter. However, calculating Equation for a situation like the jet aircraft pictured below is not practicable, and although solving Equation for a microfluidic device is doable, it is a lot of labour down the toilet. Much of computational fluid dynamics (CFD) is therefore dedicated to determining appropriate approximations to Equation in order to produce correct results at an acceptable computing cost.

Continuum Theory with Rarefied Flows

The flow equations are based on the continuum hypothesis, which states that a fluid may be thought of as a continuous entity rather than a collection of discrete molecules. Rarefied flows are those in which molecular effects are significant. The Knudsen number quantifies the degree of rarefaction:

Where the molecular mean free route and L is a length scale indicative of the flow shape, such as channel width.

$$\mathrm{Kn} = \frac{\lambda}{L}$$

A flow may be considered a continuum flow if the Knudsen number is less than 10-3. Under normal conditions, liquids and gases may nearly always be considered continuous. For gases at extremely low pressures or gas flows restricted in very tiny domains, the molecule in the fluid may interact with the same frequency as the walls that constrain the flow. The fluid flow in such systems must be represented using rarefied flow equations or, at the very least, Knudsen boundary conditions.

Non-Newtonian and Newtonian Fluids

The viscosity of a fluid is one of its characteristics. The viscous effects are represented by the viscous stress tensor. Newtonian fluids are the most often encountered fluids, including water, gases, and ethanol. The viscous stress in Newtonian fluids is proportional to the deviatoric stress tensor:

$$\tau = \mu\left(\nabla \mathbf{u} + \nabla \mathbf{u}^{\mathrm{T}}\right) - \frac{2}{3}\mu \mathbf{I}(\nabla \cdot \mathbf{u})$$

However, there are certain fluids that do not follow the basic relationship in Equation (3). These fluids are known as non-Newtonian and may exhibit a variety of behaviours. Non-Newtonian fluids include blood, paint, certain lubricants, cosmetic products, honey, ketchup, juice, and yoghurt, and various suspensions, such as sand in water or starch suspended in water.

Flow of Incompressible Fluid

If the density changes in a fluid are extremely modest, it is said to be incompressible. Under mild pressure and temperature differences, this is true for liquids (unless there are considerable temperature variations) and gases. If we ignore the heating due to viscous dissipation (so-called viscous heating) and assume that the fluid is Newtonian, we may simplify Equation (1) to:

$$\begin{aligned}
\nabla \cdot \mathbf{u} &= 0 \\
\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla(\mathbf{u}) &= -\nabla p + \nabla \cdot \left(\mu\left(\nabla \mathbf{u} + \nabla \mathbf{u}^{\mathrm{T}}\right)\right) + \mathbf{F} \\
\rho C_{\mathrm{p}} \frac{\partial T}{\partial t} + \rho C_{\mathrm{p}} \mathbf{u} \cdot \nabla T &= \nabla \cdot (k \nabla T) + Q
\end{aligned}$$

The renowned Navier-Stokes equation, named after the French physicist Navier and the Irish physicist Stokes, is the middle equation in Equation. Although Navier was the first to obtain the equations, Stokes was the first to describe the actual process underlying the viscous component, thus the name of the equations. The first equation, the continuity equation, is often included in the Navier-Stokes equations. The energy equation, as shown, has been recast as a temperature equation, which is much easier to deal with. Unless the viscosity relies on the temperature, the temperature equation is for incompressible flows that are totally decoupled from the Navier-Stokes equations.

The Navier-Stokes equations solution yields the velocity and pressure field for fluid flows with constant viscosity and density. If information on the temperature field is desired, the temperature may be solved individually. Buoyancy is a significant physical phenomena that is caused by differences in density. However, by including buoyancy as a momentum source/sink in the momentum equations, Equation may still be utilised to simulate the influence of buoyancy. Even when the density is nonconstant, the Navier-Stokes equations may be employed, and buoyancy can be added as a momentum source/sink in the momentum equations. For example, buoyancy causes cigar smoke to flow upward.

Reynolds' Number

The Reynolds number is a fundamental notion in fluid flow. It is defined as follows:

$$\mathrm{Re} = \frac{\rho U L}{\mu}$$

Where U represents a typical velocity scale and L represents a representative length scale.

In the absence of body forces and assuming constant density and viscosity, the Navier-Stokes equation (middle expression in Equation) may be nondimensionalized to read:

$$\frac{\partial \mathbf{u}'}{\partial t} + \mathbf{u}' \cdot \nabla(\mathbf{u}') = -\nabla p' + \frac{1}{\mathrm{Re}} \Delta \mathbf{u}$$

Where is a typical pressure level.

The Reynolds number, as shown in Equation (6), quantifies the relative relevance of viscous stresses. At low Reynolds numbers, the flow is totally regulated by viscous factors, but at extremely high Reynolds numbers, the flow is virtually inviscid.

Keep in mind that a given flow arrangement might have many Reynolds numbers. A channel flow, for example, might be based on either half of the channel width or the whole channel width. The velocity might be either the average or maximum velocity. As a result, knowing which length scale and velocity scale are linked with a certain Reynolds number is critical, particularly when comparing Reynolds numbers across comparable flow configurations.

The Stokes Flow

Creeping flows are defined as flows with extremely low Reynolds numbers. They are common in microfluidic systems (such as the micromixer pictured below) and lubricating systems.

$$\begin{aligned} \nabla \cdot \mathbf{u} &= 0 \\ 0 &= -\nabla p + \mu \Delta \mathbf{u} \end{aligned}$$

A fluid flow model in a micromixer.

The Stokes equations are often used to simulate flow in microfludics, such as this micromixer's flow. Stokes flow is the limit of when. Although Stokes flow may officially support both time dependence and variable material characteristics, it is written for incompressible quasistatic conditions:

The equations are named after the Irish physicist George Gabriel Stokes, who described viscous momentum transfer using these equations for the first time. The fluid determines which terms to keep in the energy equation. The convective term, as well as pressure-work effects, are often overlooked. Viscous heating is also useful for Stokes flows in bearings and other lubrication applications.

Flow in Turbulence

The Reynolds number compares the relevance of inertial effects to viscous effects. As long as the Reynolds number is not too high, viscous effects will dampen out flow field disturbances. Laminar flows are the name given to such flows. Because the viscosity dissipates any flow structures that are tiny enough, it is frequently possible to solve Equation (4) for laminar flows. The greater the Reynolds number, the greater the dominance of inertial effects over viscous effects. When the Reynolds number is high enough, any little disturbance feeds on the mean flow momentum, causing it to expand and cause new flow structures. This is referred to as transition. Turbulent flow is defined as a flow that has transitioned. Turbulent flows are distinguished by apparently chaotic eddies that cover a wide range of length scales, from massive vortices almost as big as the computational domain to microscopic dissipative eddies as small as micrometres. Because of the vast range of scales, the pure Navier-Stokes equations cannot model many turbulent flows at a reasonable processing cost. Direct numerical simulation (DNS) is achievable for certain very basic flow scenarios, but it demands massive processing resources.

$$\begin{aligned} \nabla\cdot\bar{\mathbf{u}} &= 0 \\ \rho\bar{\mathbf{u}}\cdot\nabla(\bar{\mathbf{u}}) &= -\nabla\bar{p}+\nabla\cdot\left(\mu\left(\nabla\bar{\mathbf{u}}+\nabla\bar{\mathbf{u}}^{\mathrm{T}}\right)-\rho\overline{\mathbf{u}'\mathbf{u}'}\right)+\bar{\mathbf{F}} \end{aligned}$$

We commonly use approximate turbulence models to predict the flow and pressure fields when we don't have access to a supercomputer. Turbulence models define many sorts of conservation expressions for turbulence in an averaged sense, such as looking at the conservation of kinetic energy that these tiny eddies may have (called turbulent kinetic energy). The preserved qualities, such as turbulent kinetic energy, are employed to create eddy viscosity, which is an extra contributor to viscosity. The eddy viscosity increases the viscous transfer of momentum to simulate the momentum transmitted by the small-scale eddies that we can't afford to resolve. The Reynolds-averaged Navier-Stokes models (RANS models) are the most often used turbulence models in engineering, in which the modelled variables are time-averaged and fluctuations are dealt in an added quantity known as the Reynolds stresses. RANS equations for incompressible flows are as follows where bar represents an averaged quantity and prime is the departure from the average.

For example, the unfiltered velocity may be written. When Equation (8) is compared to the continuity and momentum equations in Equation (4), the equations are identical except for the fact that unfiltered quantities have been replaced by filtered quantities, there is no time derivative (because we averaged over time), and there is an extra term in Equation (8). (8). The Reynolds stress tensor describes the influence of turbulent fluctuations on the filtered velocity and pressure fields. It is feasible to write transport equations for the Reynolds-stress tensor entries. These equations, with proper simplifications and assumptions, may provide so-called Reynolds-stress models. While powerful, these models are often difficult to work with, and even though they are far less costly computationally than DNS, they are still too expensive for most industrial applications.

Instead, consider that turbulence serves as an extra viscous effect and write, where is the turbulence viscosity, also known as the eddy viscosity. The most extensively used eddy viscosity models for industrial applications are k-, k-, shear stress transport (SST), and the Spalart-Allmaras turbulence models. Another kind of turbulence model averages turbulence across a limited geographical area rather than throughout time. This creates a low-pass filter for eddies of a specific length scale. Big turbulent eddies are resolved in this manner, whereas the influence of minor eddies must be considered, thus the phrase large eddy simulation. For incompressible LES, the continuity and momentum equations are the same:

The term is the subgrid stress (SGS) tensor and it shows the influence of the subgrid scales on the resolved scales. A widely used model for the SGS tensor is the Smagorinsky-Lilly model. LES is generally more accurate than RANS, but the simulations must always be 3D, even though the flow is really 2D and the simulations are always time dependent. In addition, the needed resolution for the SGS models to be valid is frequently fairly high, which implies that LES is only utilised when even the most powerful RANS models fail to capture the fundamental aspects of the A model illustrating turbulent wind flow around a solar panel. Turbulent wind flow around a solar panel. The low-Reynolds RANS turbulence models may be used to determine the force exerted by the wind on the panels.

The Mach Factor

The Mach number is defined as follows:

$$\mathrm{Ma} = \frac{|\mathbf{u}|}{c}$$

Where c is the sound speed.

The Mach number compares the speed of a fluid to the speed of pressure waves. When the Mach number is tiny, that is, when the pressure waves are so rapid that they essentially reduce to a mass conservation constraint, the pressure waves are said to be fast. Formally, the incompressible flow formulation in Equation (4) may be obtained by allowing. As the Mach number approaches one, or as the velocity approaches the speed of sound, the effects of pressure waves must also be considered. Because viscous heating is often essential in these circumstances, we must solve the whole set of continuity, momentum, and energy equations provided by Equation. Compressible viscous flow is defined as flow in which all components in Equation are significant.

When the Mach number is high, the Reynolds number is generally high as well, since both are related to the velocity. As a result, Equation is often supplemented with a turbulence model to account for momentum eddy diffusivity for heat transmission. Equation and its turbulence model often have a significant connection.

A compressible turbulent flow model.

The k-turbulence model was used to simulate fully compressible turbulent flow. The diamond-shaped pattern of the velocity field induced by the pressure shocks can be seen (shock diamonds).

The Inviscid Flow and Euler Equations

For the flow of gases at moderate pressures at and above the speed of sound, the contribution of molecular viscosity and eddy viscosity to momentum transfer is often overlooked. In such instances, the model equations explain momentum conservation (without a viscous factor), mass conservation, and energy conservation.

There is no requirement for a turbulence model since eddy viscosity is not taken into consideration. Heat transmission by conduction is the energy equation's counterpart to viscous momentum transfer. In reality, the same process that causes viscosity also causes thermal conductivity, and the eddy diffusivity for momentum transfer is utilised to calculate the eddy diffusivity for heat transmission. As a result, in circumstances where we can disregard viscous momentum transfer, we can typically disregard heat transmission through conduction in the energy calculations. The conservation equations for inviscid flow and negligible thermal conductivity are often known as the Euler equations, after the eminent Swiss mathematician who developed them. The Euler equations are as follows:

A supersonic flow model in a bumpy channel.

This benchmark issue for the solution of the Euler equations for high Mach number flow involves supersonic flow over a wing-shaped obstruction that creates pressure shocks that reflect on the walls.

Flow in Multiple Phases

The equations for momentum, mass, and energy conservation may also be used to fluid flow including several phases, such as a gas and a liquid phase or two separate liquid phases, such as oil and water. Surface tracking techniques, such as the level set or phase field approaches, provide the most precise representation of multiphase flow. Surface tension, for example, is incorporated as a source or sink in the momentum equations at a thin layer with a very tiny thickness that follows the boundary between the phases in these models. The phase boundary's form and location are calculated in great detail. This implies that the momentum and mass conservation equations are paired with a set of transport equations for a level set or phase field function that describes the location of the phase boundary at a given value (isosurface).

Traditional benchmark model for two-phase surface tracking flow models. During sloshing, the heavier fluid becomes connected to the upper wall for a very brief duration. Surface tension is responsible for this adhesion.

We cannot trace the form of a phase boundary when it consists of millions of droplets or bubbles, or when the structure of the phase boundary is exceedingly complicated in its features. Instead, we must homogenise the data and consider the existence of distinct phases as fields of averaged mass or volume fractions. We no longer keep detailed track of the phase boundary shape. Instead, we characterise the potential interactions between the phases as momentum sources and sinks distributed across the fluid mixture. Furthermore, the momentum and mass conservation equations are coupled with a transport equation for the volume fraction of one of the phases in the case of two-phase flow and two transport equations in the case of three-phase flow. When the density difference between two phases is considerable, we may need to develop distinct momentum equations for each phase specified everywhere in the fluid domain.

A dispersed flow model, such as the bubbly flow model, may provide a suitable depiction of the homogenised two-phase flow for bubbles in liquids. We may use a somewhat more sophisticated model, such as the mixture model for multiphase flow, for liquid-liquid mixtures such as oil and water. A liquid-liquid extraction column model. The graph depicts the volume percent of oil. The heavier water solution comes in at the top annular intake, while the oil phase comes out through the top circular outlet. When there are a high number of solid particles in a gas and the density difference is quite great, we often need to write momentum equations for both the dispersed solid particles and the gas phase. Euler-Euler multiphase flow models are models that specify the momentum equations for each phase. The term derives from the fact that both phases are characterised as continuous, i.e. using an Eulerian method.

When the particles are few enough, a particle tracking approach may be used to characterise the dispersed phase. The Euler-Lagrange technique is so named because the continuum (for example, the fluid) is described by an Eulerian approach, while the particles are represented by a Lagrangian approach. The Euler-Lagrange technique has the benefit of allowing attributes to be connected with each individual particle, but it gets highly costly as the number of particles rises. The distinction between separated multiphase flow models (left) and scattered multiphase flow models (right) (right). The phase boundary in the surface tracking technique is represented by the isosurface of the field at = 0. Only the volume fraction of bubbles or droplets is obtained in the dispersed multiphase flow model, while the features of the phase boundary are regarded as averaged volume forces.

Flow of Porous Media

If we can afford to describe a porous structure in detail, including all of its surface structures and surface properties, we can use the equations for momentum and mass conservation to define no-slip conditions on the pore walls or the Knudsen condition if the mean width of the pores is of the same order of magnitude as the scale of molecule interactions.

However, in most circumstances, we cannot afford to represent the millions of pore bends and structures in a porous structure's macroscopic model. As a result, porous media flow models often use homogenization to characterise the fluid and porous matrix domains in the porous structure as a slab with averaged attributes such as averaged porosity, tortuosity, and permeability. The momentum equations are then transformed into Darcy's law, named after the French engineer who initially proposed it.

Hydrodynamics

Bernoulli’s law

Until date, the emphasis has been on fluids at rest. This section is concerned with fluids that are in constant motion, so that the fluid velocity at any given place in space does not change over time. Any steady flow pattern may be represented by a series of streamlines, which are the trajectories of fictional particles floating in the fluid and transported along with it. The fluid is moving in constant flow, but the streamlines are stationary. The fluid velocity is quite high when the streamlines pack together; where they widen out, the water becomes comparatively stagnant. When Euler and Bernoulli established the foundations of hydrodynamics, they viewed the fluid as an idealised inviscid material in which the shear stresses associated with viscosity are 0 and the pressure p is isotropic, as in a fluid at rest in equilibrium. They discovered a simple rule that relates the variation of p along a streamline to the fluctuation of v (the principle is attributed to Bernoulli, although Euler seems to have discovered it earlier), which helps to explain many of the phenomena seen in actual fluids in steady motion. There is no one solution to the inevitable issue of when and why it is acceptable to ignore viscosity. Some responses will be offered later in this report, but other issues will be addressed first.

Consider a tiny element of fluid of mass m that, apart from the force of gravity, is exclusively acted on by a pressure p. The latter is isotropic and does not change with time, although it may vary from one place in space to another. It is a well-known result of Newton's laws of motion that when a particle of mass m moves under the influence of its muscle mass mg and an additional force F from a point P where its speed is vP and its height is zP to a point Q where this speed is vQ and its height is zQ, the work done by the additional force equals the particle's increase in kinetic and potential energy. In fluid dynamics, Bernoulli's theorem describes the relationship between pressure, velocity, and elevation in a flowing fluid (liquid or gas), when compressibility and viscosity (internal friction) are minimal and the flow is steady, or laminar. The theorem, which was first derived by the Swiss mathematician Daniel Bernoulli, states that the total mechanical energy of a flowing fluid, which includes the energy associated with fluid pressure, the gravitational potential energies of elevation, and the kinetic energy of fluid motion, remains constant. Bernoulli's theorem is a concept of energy conservation for perfect fluids in steady, or streamline, flow that serves as the foundation for numerous engineering applications.

According to Bernoulli's theorem, if the fluid flows horizontally with no change in gravitational potential energy, a reduction in fluid pressure is related with an increase in fluid velocity. If the fluid flows through a horizontal pipe with varied cross-sectional area, the fluid speeds faster in restricted sections and exerts the least pressure where the upper part is smallest. The Venturi effect is named after the Italian scientist G.B. Venturi (1746-1822), who first observed the effects of confined channels on fluid flow.

$$\int_{\mathrm{P}}^{\mathrm{Q}} \boldsymbol{F} \cdot d\boldsymbol{s} = \left(\frac{1}{2}\right) m(v_{\mathrm{Q}}^2 - v_{\mathrm{P}}^2) + mg(z_{\mathrm{Q}} - z_{\mathrm{P}}).$$

According to Bernoulli's law, if an inviscid fluid flows down a pipe with changing cross section, the pressure is comparatively low at constrictions where its velocity is high and relatively high when the pipe expands out and the fluid stagnates. When many individuals first experience this condition, they find it perplexing. They argue that a constriction should raise rather than decrease local pressure. The paradox dissolves when one learns to consider pressure changes along the pipe as causes and velocity changes as effects, rather than the other way around; it is only because the pressure drops at a constriction that the pressure luminance upstream of the constriction has the right sign to cause the fluid to accelerate.

Whether paradoxical or not, predictions based on Bernoulli's rule have been thoroughly tested through experiment. Try holding two pieces of paper vertically two cm apart and blowing downward to create an air current between them. The drop in pressure caused by this current will bring the sheets closer. Ships are pulled together for similar reasons if they are travelling in the same direction in the same speed with a little space between them. The current in this situation is caused by the displacement of water by each ship's bow, which needs to flow backward to fill the space produced as the stern goes forward, and the velocity between the ships, to which both contribute, is greater than the current passing past their outer edges. Listen to the hissing sound created by a tap that is nearly but not quite turned off as another basic experiment. In this scenario, the flow is so restricted and the velocity so high inside the constriction that the pressure within the contraction is actually negative. The water cavitates as it flows through, aided by the dissolved gases that are generally present, and the noise heard is the sound of small bubbles bursting as the water starts to slow down and the pressure increases again on the other side.

CHAPTER 6

MOMENT OF FORCE

Mr. Alok Raj,
Assistant Professor, Department of Chemical Engineering, Jaipur National University, Jaipur India
Email Id- aloksraj06@jnujaipur.ac.in

In physics, the moment of force, or simply moment, is a measure of the propensity of a body to rotate around a fixed point or axis.The moment arm is described in this idea as the distance from the of rotation. This distance is very crucial. By altering the distance, i.e. the moment arm, the lever, pulley, gear, and most other basic machines provide mechanical advantage. According to the Principle of Moment, when a system is in equilibrium, the total of its Clockwise moments equals the sum of its Anticlockwise Moments. Moments, or turning effects, are used in a variety of applications, including seesaws, opening and shutting doors, nutcrackers, can opener, and crowbars.

The Moment Formula

A lever, as we know, is a basic mechanism in which one force, known as the effort, is utilised to overcome another force, known as the load. As a result, a moment in physics is a combination of a real number and a distance. The newton metre (kgm2 per s2) is the SI unit for a moment. In addition, the moment of force is expressed in Nm.

Notch and its types in fluid mechanics

Notch and weir

A notch is a hole created in a tank or other vessel's wall such that the liquids surface lies below the opening's top edge. A weir may be described just like any regular obstacle in open stream on which the flow occurs. An impediment that allows liquid to flow over it is known as a notch. The notch functions as a helpful measurement instrument since the discharge is connected to the coefficient of discharge far above floor of the notch.

The aperture is placed at the side of the tube in the case of a measuring storage reservoir and as such the liquid surface is below the outer side of something like the opening. In actuality, this is a sizable aperture with no top boundary, meaning that its size varies according to the altitude of the outer surface. Weirs are large scale notches used to gauge the flow of rivers, canals, etc. It is a wide structure made of masonry or concrete that spans the river there in flow direction. As a result, the extra water can flow towards the downstream side over the whole length of the pipe. Thus a weir is similarly to a tiny dam placed across the riverbed, with just a distinction that the surplus water flows upstream only through a short piece called spillway while in case of weir, this same excess water flows across its full length

Nappe and crest

The term nappe or vein refers to the water sheet that passes through with a notch or over a weir. Sill or crest refers to the top of a weir or the bottom edge of a slot over which water flows. The crest height is the amount above the tank or channel's bottom. A nappe is a sheet or veil of rainwater that passes over a weir or dam in hydraulic engineering. The crest of something like a dam or weir creates distinct features on the top and lower water surface

Types of notch

Depending on their shapes, notches come in a variety of forms. However, the following are significant from the perspective of the topic.

1. Rectangular notch
2. Triangular notch
3. Trapezoidal notch
4. Stepped notch

Rectangular notch

A typical tool used to control and measure outflow in irrigation projects is the rectangular weir (notch). The current study focused on laboratory tests looking at the hydraulic properties of rectangular notches.

Triangular notch

A triangular notch or V-notch is another name for a triangular notch. Think of a triangular slot on the tank's side where water is flowing. Let H be the height of the liquid above the notch's peak. = The notch's angle

Trapezoidal notch

A trapezoidal notch is made up of two triangular notches and a rectangular notch, the discharge over such a notch will thus undoubtedly equal the total of the discharges over the rectangular and triangular notches, it follows.

Stepped notch

A stepped notch is made up of many rectangular notches. Therefore, it follows that the discharging over a notch of this nature will consist of the total of both the discharges over the several rectangular notches. An intentionally created V-shaped, U-shaped, or semi-circular flaw in a planar material is referred to as a "notch" in mechanical engineering and physical science. A notch in a structural component creates a stress concentration that may lead to the beginning and development of fatigue fractures. To assess parameters related to fracture mechanics, such as micro hardness and rates of cyclic loading, notches are applied in materials characterisation. When a controlled origin morphological crack is required for a standardized evaluation of the material's fracture resistance, notches are frequently employed in material impact testing. The most popular one is the Impact test test, which involves striking a horizontally notched specimen with a pendulum hammer (striker).

Application of notch

Tanks, reservoirs, or indeed any type of water storage system with a path for water to leave are examples of applications for notches. Similar to a notch on a big scale, a weir is used to gauge the flow of canals and rivers. Notch and weir are further utilized in addition to measurement to control the discharge of small and large channels. A notch is a type of fluid flow aperture that extends across its breadth and allows fluid to pass through the bottom. It is employed to control fluid flow. The bottom end of the notch is often sharpened to provide the least amount of resistance to fluid passage (generally water). A notch is often constructed of a steel plate and used to control small-scale fluid flow. The stopband refers to the frequency range that a music group filter attenuates. A notch filter gets its name from the fact that its narrow stopband causes the frequency response to resemble a deep notch. Additionally, it indicates that the Q factor, or the proportion of the center frequency to the bandwidth, of notch filters is large.

An aperture created in a tank's side that extends even beyond the liquid's free surface is referred to as a "notch." It resembles a big hole with no upper edge in several ways. Typically, a notch is used to gauge a tank's water flow. Although created on a much larger scale, a weir is similar to a notch. The weir is a slot carved in a dam used to release extra water. Water goes through an aperture while flowing over just a notch or weir. While the sheet of water produced by a fissure or weir is known as a nappe, the body of water discharged via an orifice is known as a jet.

Boundary Layer in fluid mechanics

Boundary Layer

A boundary layer in fluid mechanics is a thin sheet of a flowing gaseous state in contact with an object such as the inside of a pipe or perhaps the wing of an aircraft. Shear forces are exerted on the fluid inside the boundary layer. As long as even the fluid is in contact with water, a spectrum of velocities exists over the boundary layer, ranging from maximum to zero. The boundary layers of an aircraft wing are thinner at the forefront and thicker so at trailing edge. Such boundary layers frequently contain turbulence in the point of discharge and working fluid in the upstream or leading section. Also see turbulence and laminar flow. A boundaries layer is the layer layer of liquid fluid formed when fluid flows along a surface and surrounding a bounding surface. As a product of the fluid-wall interaction, some no boundary condition is formed. The airflow above the surface then gradually increases until it approaches the bulk rate of flow again. The phrase "flow boundary layer" applies to thin fluid layers that have not yet recovered to the velocity of the main flow. Figure 6. 1: illustrates the laminar boundary layer flow and Turbulent boundary layer

Types of Boundary layer

There are two types of boundary layer which are demonstrated below:

Laminar boundary layer flow

The laminar flow produces less skin friction drag than just the turbulent flow, but it is less stable. The laminar boundary is a highly smooth flow, whereas the turbulent boundary layer comprises swirl or "eddies. Boundary layer flow begins as smooth spontaneous convection over a wing surface. As the flow flows away from the leading edge, the layer of laminar boundaries thickens.

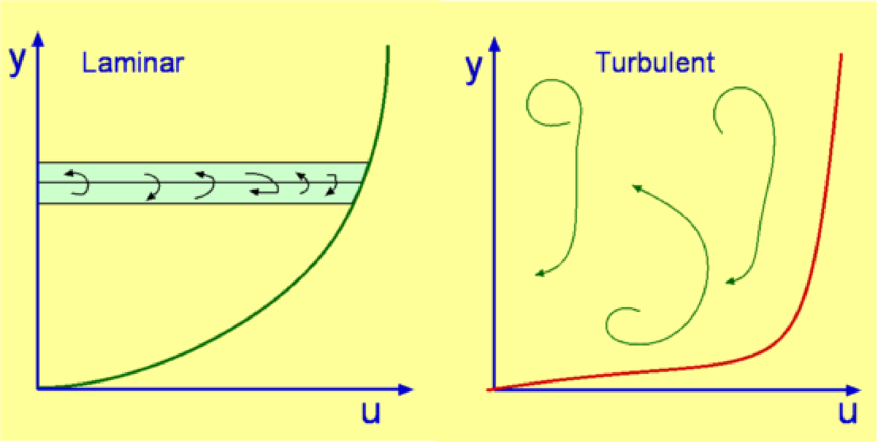

Figure 6. 1: illustrates the laminar boundary layer flow and Turbulent boundary layer.

Laminar boundary layers are also another possibility. When fluid passes in layers, all layer passes through the layer upon layer around it, forming a boundary condition layer. This is in contrast to the highly agitated Turbulent Boundary Layers seen in Fig.1. Any momentum or momentum transmission in a boundary occurs only on a microscopic scale that is imperceptible to the human eye. Shear stress may therefore be calculated using molecular viscosity. Laminar boundary layers arise only once the N values have decreased.

Turbulent boundary layer flow

The smooth laminar flow degrades and changes to a turbulent flow a certain distance behind the leading edge. A considerable section of the wing surface should be within the laminar component of the boundary layer, or the changeover from scene to scene flow should occur as far aft here on wing as practicable. But compared to the turbulent layer, the low energy fluid motion usually fails more abruptly. The mixing over many levels of a boundary layer, on the other hand, indicates a turbulent boundary layer. Now the mixing has reached the macroscopic level. Fluid may be observed travelling across in packets. Therefore, compared to the laminar boundary layer, there is a far larger interchange of mass, momentum, and energy. Only at higher Reynolds numbers can a boundary layer separation actually develop. Molecular viscosity alone cannot manage the magnitude of mixing. The term "turbulence viscosity" or "eddy viscosity," which lacks an exact meaning, is used to calculate turbulent flow. A model must be created. For this, numerous models have been created.

A layer of rotating fluid, the boundary layer, arises in consequence to the action of viscosity but the no support conditions on the surface when a traveling viscous fluid comes into contact with the material surface. By first simplifying this same Navier-Stokes equations to take into account the geometry of the flow, and then finding solutions the simplified equations, the flow within this layer can be determined when the surface is flat or slightly curved, and the pressure gradient that forms on it is thin and stays adjacent to it. Unfortunately, the boundary layer may depart the surface when the pressure rises in the downstream or when the surface is extremely curved.

The distance from the rigid body to the point where the viscous flow velocity is 99% of the aerodynamic force (the surface velocity of an inviscid flow) is the conventional definition of the thickness of the fluid velocity. Displacement surface area is an unconventional definition that claims the shear layer represents a deficit in flow velocity compared to an inviscid flow with slip at the wall. It is the amount of wall displacement required in the incompressible viscous case to produce a total mass flow equal to that in the viscous case. The no-slip requirement demands that the fluid temperature be the same as the surface temperature and that the flow rate at the surface of the solid object be zero. The distance from the body over which the temperature is 99% of the flow velocity temperature is also known as the thermal boundary layer thickness. The Prandtl number determines the proportion between the two thicknesses. The two field lines have the same thickness if the Prandtl is 1. The thermal boundary layer is narrower than the velocity of the fluid if the Prandtl number is higher than 1. The temperature boundary layer becomes thicker than the limit layer if the Local nusselt number is less than 1, which it is for air under typical conditions.

Impact of Jet in Fluid mechanics

Jet in Fluid Mechanics

A jet is a stream of fluids that also is discharged into the environment, often via a nozzle, aperture, and orifice. Jets may travel very far without losing energy. When compared to the underlying fluid medium, jet fluid has more momentum. Entrainment is the process by which the surrounding fluid is transported with the jet when the surrounding material is considered to be composed of the same fluid as the jet and this fluid has a viscosity

Impact of jet

The impact of jet consisted of transparent type of cylinder container containing a vertically tapered nozzle and test plate. The liquid emerges as a jet from a nozzle's output that is attached to a pipe. A jet is a stream of water that is pushed into a space through which liquid is moving under pressure. Jets can flow through a surrounding medium, often through some type of nozzle, aperture, orifice. Without dispersing over a great distance. When compared to the adjacent fluid medium, jet fluid has more momentum. The scenario when the Assuming that the fluid in the surrounding medium is the same as that in the jet, this fluid possesses Entrainment is the process by which the adjacent fluid is transported with jet in the presence of a viscosity.

A water jet that emerges from a pump has a velocity, which means it contains kinetic energy. It is stated to have an influence on the plate if this jet collides with it. On the plate it impacts, the jet will apply force. Dynamic pressure applied by the jet is the name given to this force. This force results from the jet's altered momentum as a result of the collision. This force is equivalent to the rate at which momentum changes, or (mass impacting the plate each second) x (change in velocity).

Straight Impact of a Jet on a Motionless Flat Plate:

A regular water jet might impact a flat plate.

Let,

The jet's cross-sectional area in meters2.

V = Velocity of the jet in meters per second.

M = Mass of water striking the plate per second.

∴ M = ρaV kg/sec

Where ρ = density of water in kg/cum

The jet's force on the plate is-

P = Momentum Change/Second.

= mass per second that the plate is struck (Change in velocity)

= MV = aV.V = M (V - 0).

Slanting Impact of a Jet on a Motionless Flat Plate:

After contact, the velocity component perpendicular to the plate is zero

The force that the jet applies in a normal towards the plate

P is equal to (Mass striking the plate per second) x. (Change in velocity normal to the plate)

P = M = a V. V sin (V sin - 0)

Newton's formula is P = aV2

Because the point of action of the force does not shift in the two situations outlined above, the work performed by the stream on the plate is zero.

Straight Impact of a Jet on an Affecting Plate

Let,

V is the jet's speed

v is the plate's velocity.

Jet velocity in relation to plate velocity = (V – v)

May assume that the plate is at stationary but that the jet is travelling away from the plate at a velocity of (V -v).

Force applied to the plate by the jet

P equals a (V - v)2 Newton.

The point of action of the force is moving in this instance, hence the jet is doing work.

Per second, the jet produces work on the plate

Pv is equal to (V - v)2 v Nautical miles or Joule/sec.

Oblique Impact of a Jet on a Moving Vane

Let V and v be the jet's and the vane's respective velocities in the very same direction. Let there be an angle of at which the jet meets the plate. In this instance, the speed at which the liquid strikes the plate

$= \rho a (V – v) (V – v)$

Prior to collision, relative velocity equal to the plate

$= (V - v) \sin \theta$

After collision, the relative velocity along the plate's normal was 0.

The force that the jet applies in a normal to a plate

$P = a [(V - v) \sin - 0] (V - v)$

The components Px and Py in the plate's motion direction and perpendicular to the plate's Motion may be separated from the force P acting perpendicular to the plate.

Nozzle and Nozzle Wear

Nozzle

A nozzle is always a pipe of tube with a variable cross sectional area that is used to control, direct, or alter the flow of liquid (liquid or gas). Nozzles are widely employed to regulate the stream that flows from them in terms of its mass, shape, speed, direction, pressure, and/or rate of flow. The melt can exit the barrel and enter to mound through the nozzle, which is fitted into the barrel of the gun. Additionally, it is a location where the melt may be warmed by friction and conduction before entering the mold's relatively cold channel. When the nozzle comes into contact with the mold, heat is transferred from the nozzle; however, if this heat transfer is considerable, it is best to remove the nozzle first from mold as during screw-back phase of a molding cycle. If not, the plastic might freeze inside the nozzle

Nozzles are contoured conduits used to accelerate liquids or gases to a desired speed in a predetermined direction. Nozzles are employed in gas dynamic lasers, gas turbines, jet devices, intense shattering or spraying technologies, rocket and aviation engineering to create jet propulsion (see Gas turbine). The fundamental building blocks of wind tunnels (see Wind Tunnels) are nozzles, which enable the construction of systems for intense (jet) cooling, like in electrical engineering

Types of Nozzle

The types of nozzle is described into four categories:

1. Jets nozzles
2. High velocity nozzles
3. Propelling nozzles

4. Spray nozzles

Jets Nozzles

A nozzle used to propel gas or liquid into the environment in a cohesive stream is known as a jet, fluids jet, or hydro jet. Gas jets are frequently seen in gas stoves, ovens, or grills. Prior to the invention of electricity, gas jets were a prominent source of illumination. In carburetors, where smooth, calibrated orifices are utilized to control the flow of gasoline into an engine, there are additional varieties of fluid jets

High velocity nozzles

For usage in fixed wash water or inundation systems for fire prevention applications, high velocity water sprays nozzles are external swirl plate kind open nozzles. To put out fires, these nozzles provide a solid, consistent, and dense center of high-velocity water spray.

Propelling nozzles

A variable area propelling nozzle through its open configuration is used by engines that need to generate thrust fast, from idle, in order to minimize thrust and maintain high engine rpm. A propelling nozzle is a part of a jet engine that is located at the tail of something like the engine and operates to form a combustion jet and to maximize the use of the exhaust stream energy throughout the engine. Closing the nozzle toward the powerful position is easy and quick when propulsion is required, such as when starting a go-around.

Spray nozzles

A basic tool called a spray nozzle is used to divide a water dynamics into a spray nozzle. Despite the nozzles' apparent simplicity, there are many distinct items in our assortment that represent the many ways that diverse sectors require to spray different fluids.

Nozzle Wear

The nozzle's wear mechanism is comparable to those of other conventional machining operations that employ cutting tools. The nozzle wall may deteriorate due to the water's high pressure and harsh abrasive particles. Fiber reinforced polymer is a material that may be used with 3D printers more and more often. Carbon fiber is the most used type of filler. With more people using 3D printing, fiberglass is moving up to second. Long before 3D printing, it was usual in injection molding. These extra fibers are intended to improve the mechanical qualities. In general, see a rise in bending modulus and tensile strength. These polymer and fiber blends are referred to as "Filled Plastics" and "Composites" at times.

Nozzle wear falls into one of two categories:

Breakage of nozzle body

The nozzle's spray pattern, pressure, and flow will all be subpar. Mechanical harm caused accidentally. The most frequent issue resulting in subpar spray nozzle performance and nozzle replacement is this one. Poor spray patterns, higher pressure, and lower flow are the results. The throat Mach number cannot rise over one even after boosting the nozzle pressure ratio further. The flow is ready to develop to supersonic speeds downstream (i.e. outside the nozzle), however Mach 1 can be a high speed for a hot gas since the speed of sound changes with the squared of absolute temperature. This knowledge is often applied in the rocketry industry, where hypersonic velocities are necessary and certain propellant combinations are utilized to further boost sonic speed .Divergent nozzles speed up sonic or hypersonic fluids while slowing subsonic fluids. After making contact with the coal's surface, the minute water droplets will begin to move laterally under a specific amount of pressure.

Water is driven to enter the pores and fractures of the coal body then interact with them because of the pressure differential between the liquid water and the hole fluid on the inside of the coal species, which is similar to imparting a quasi-static pressure on the coal body. When the contact reaches critical force for fracture propagation, the pores and cracks would keep growing and converge to generate a secondary failure. Continuous high-pressure liquid passing through the spray gun opening over time is what causes erosion. The flow widens the aperture while progressively removing metal. The outcome is an uneven spray pattern, an increase in flow, a drop in pressure.

Turbine and its concept

Turbine

Turbine is a machine which takes the angular momentum of fluids and turns into mechanical energy throughout a specified mechanism. The turbine consist of many blades which are attached to an axle and is used typically to operate a generator. A turbine is a rotating mechanical device that takes energy from the fluid flow and transforms it into productive work (from the Greek, tyre, or Spanish turbo, meaning vortex). When used in conjunction with a generator, the work that a turbine does may be utilized to create electricity. A turbine is a components of performance having at least single moving component called a rotor component, which is a shafts of drum with connected blades. The blades are affected by a moving fluid, which causes them to move and give the rotor rotational energy. Waterwheels and windmills are two early turbine types

Hydraulic Machines

The term "hydraulic machines" refers to any device that transforms hydraulic energy (the energy contained in water) into mechanical energy, which is then transformed into electrical energy, or mechanical power into hydraulic energy.

Turbines are hydraulic devices that transform hydrodynamic energy into mechanical energy. By definition, turbines are hydraulic devices that transform hydraulic energy into mechanical energy. An electric generator that is directly connected to the turbine shaft uses this mechanical energy to run. This results in the conversion of mechanical energy to electrical energy. Hydro-electric power is the name given to the electric power produced by hydraulic energy (water energy)

Types of Turbine

The turbine are of four types

1. Steam Turbine
2. Vapor Turbine
3. Gas Turbine
4. Hydraulic Turbine

Hydraulic Turbine

A rotating device known as a hydraulic turbine or turbine shaft uses the potential and kinetic energy of moving water to produce mechanical work. The primary device that aids in converting hydrostatic energy into electricity with the aid of a generator is a hydraulic turbine. The turbine is forced to revolve when a jet of water strikes its blades; thus, the turbine is equipped with a generator, which likewise rotates and generates electrical energy

Vapor Turbine

A steam turbine is a device that uses pressurized steam's thermal energy to drive mechanical energy on a revolving output shaft. Charles Parsons created it in its current form in 1884. Modern steam turbines are made utilizing sophisticated metalworking techniques that were first made possible in the 20th century. The continuing improvement of steam turbines' resilience and efficacy is still essential to the economics and finance of the 21st century.

Gas Turbine

A steam turbine operates by heating water to extreme temperatures until it is transformed to steam using a heating element (gas, coal, nuclear, or solar energy). That steam begins to cool as it passes through the turning turbine blades. Thus, in the rotating turbine's blades, the steam's potential energy is converted to kinetic energy. Steam turbines are particularly well adapted for operating electrical generators for the creation of electrical power since they produce rotational motion. The generator, which has an axle connecting it to the turbines, generates energy by creating a magnetic field that induces an electric current

Steam Turbine

A steam turbine is a device that uses pressurized steam's thermal energy to drive mechanical work on a revolving output shaft. Charles Parsons created the steam turbine as we know it today in 1884. Fabricating a modern steam turbine requires advanced metalwork to shape high-grade steel alloys into precise parts using technologies who first became accessible in the 20th century; steam turbines' ongoing improvements in robustness and efficiency remain crucial toward the energy economics of the 21st century. The utilization of many stages in the expansions of the steam, and results in a review of current to the possible responses expansion process, is a major contributor to the steam turbine's gain in thermodynamic efficiency.

Application of Turbines

One of the tools that engineers utilize the most frequently today is the turbine. They are spinning devices with a variety of uses, spanning hydroelectric power to racing, as well as many purposes. Turbines are used in nuclear, hydro, thermoelectric, and other energy plant types. Wind turbines are also receiving even more investment and use in this industry. They are essentially motor machines since the working fluid moves them. For instance, the idea behind counter-working pumps. The working fluid is energized by pumps, which raises its pressure and encourages fluid flow in the pipeline.

Each of these models has a bladed rotor, which makes it distinct from other designs. Except perhaps wind turbines, it is built with nozzles, the fixed blades that are in charge of guiding the fluid toward the rotating blades. At this point, the fluid's enthalpy1 is converted into mechanical power, which causes an axis to rotate. This axis is connected to power generator in power generation turbines, which facilitates the production of electrical energy. The air that moves through the turbines used during aviation undergoes compression accompanied by combustion before exiting the device, creating the thrust required to operate an aero plane.

They are often operated with saturated steam with superheated steam inside of plants, where each application mostly maintains the machine's constructive structure. Turbine discharge pressure is a significant and often used classification. Backpressure turbines are turbines with discharge pressures higher than atmospheric. The turbine is referred to as condensation if the outlet pressure is reduced than pressure and temperature. Condensing turbines are typically big turbines that use superheated steam to function. The steam expands through multiple stages before condensing to produce just water at the conclusion of the operation.

Centrifugal Pump and its specification in Fluid Mechanics

Centrifugal Pump

Centrifugal pumps carry fluids by transferring rotational kinetic energy into water circuit flow. Rotational energy is often generated by a turbine or electric engine. The fluid passes the pump impeller along or towards the rotating axis and is driven either by turbine before flowing outwards into a distributor with volute compartments (casing) and exiting. They belong to the dynamic axial effort turbomachinery class. A centrifugal pump moves a fluid by transferring rotational momentum by one or more machine systems to one or perhaps more driven rotors known as impellers. Fluid is propelled out of the rapidly rotating propeller along its circle by centrifugal force through the vane tips. The pump output is achieved by the fluid going at a greater speed and pressure due to the functioning of the impeller. The flowing fluid is directed by the impeller into the pipe wall, to be confined, slowed, and regulated before being expelled.

Centrifugal pump working

The impeller of a centrifugal pump is its most significant component. It is composed of many bent vanes. They are usually situated between the two plates (an enclosed impeller). For water containing entrained materials, an open closed tractor trailers impeller should have been employed. Figure 6.2 Centrifugal pump working (Figure 6.2).

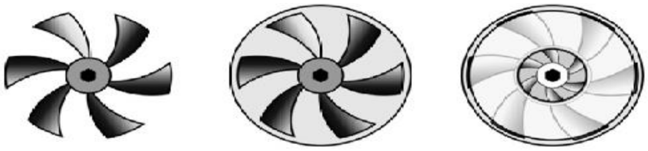

Figure 6.2 Centrifugal pump working.

The impeller's axis (the "eye") receives fluid, which departs along the circle between the vanes. The impeller is attached to a motor and rotates quickly on the side opposite the eye by a driving shaft (typically 500-5000rpm). The fluid is accelerated through to the impeller vanes and into the suction port by the impeller's rotating motion. Pump casings come in two different fundamental types: volute and diffuser. Both designs' objectives are to convert fluid flow into a regulated discharge at pressure. The impeller is offset in a volute casing, essentially forming a curved vortex with a growing cross-sectional area as it approaches the pump outlet.

Types of Centrifugal pump

Positive displacement pumps and dynamic (centrifugal) pumps are the two main categories into which pumps are often divided. In order to make the fluid flow, positive displacement pumps employ a mechanical method to change the size of (or move) inner fluid chamber. By revolving impellers that are submerged in the fluid, centrifugal pumps, on the other hand, provide the fluid momentum. At the pump output, the momentum results in a rise in pressure or flow Centrifugal pumps have changing torque characteristics, whereas hydraulic systems have a torque control characteristic. The only pumps covered in this article are centrifugal ones. The two main types of pumps are rotational and positive movement pumps. Flow rates, as opposed to reciprocating pumps, are frequently built for higher fluxes and pumping liquids with droplet diameters as small as 0.1 cP. In some chemical plants, centrifugal pumps may account for 90% of all pumps. Pumps with positive displacement, however, are favored for a few of applications.

The amount of kinetic energy a liquid has when it leaves an impeller is controlled by adding resistance to the flow. The pumps volute (casing), which captures the liquid that slows it down, produces the initial resistance. Several of the angular momentum of the liquid is turned into pressure energy when it slows down inside the pump casing. A pressure gauge connected to the discharge pipe measures the flow resistance to the pump. A pump merely provides flow; it does not create pressure. The measure of flow resistance is pressure

Limitations of a centrifugal pump

To work properly, the impeller of a centrifugal pump must continually revolve at a high speed. With very viscous inputs, centrifugal compressors lose efficiency because there is greater opposition and more suction is needed to keep a certain flow rate. As a result, centrifugal pumps are frequently used to pump fluids with viscosities ranging between 0.1 and 200 cP at moderate pressure and large capacity. High viscosity oils or slurries like mud can cause increased wear and overheating, which can result in damage and early failure. Positive displacement pumps are less likely to experience these issues since they frequently run at significantly lower rates.

Most low pressure, high capacity transferring applications using low viscosity fluids, such as water, solvents, chemicals, and light oils, may be solved simply and affordably using centrifugal pump designs. The transfer hydraulic chemicals in petrochemical facilities, irrigation, and supply of water and circulation are typical uses. For applications requiring very viscous fluids like thick oils & slurries, especially with high pressures, for complicated feeds like emulsions, meals, or biological fluids, and where precision dosing is essential, compressors are favored.

A pneumatic system, like other pumps, converts rotational energy, often from a machine, into power generation in a flowing fluid. A part of the cost is transformed into kinetic energy in the fluid. The casing's eye acts as the axial entrance point for fluid, which is then trapped in the propellers and whirled tangentially and radially outwards until it escapes via all circumferential sections of the impeller and enters the diffused portion of the casing. The fluid rises in velocity and pressure as it flows past the impeller. The casing's wedge diffuser, commonly known as the scrolling part, slows the flow while increasing the pressure.

CHAPTER 7

RECIPROCATING PUMP AND ITS SPECIFICATION

Mr. Alok Raj,
Assistant Professor, Department of Chemical Engineering, Jaipur National University, Jaipur India
Email Id- aloksraj06@jnujaipur.ac.in

A reciprocating pump is a radial flow pump in which a predetermined amount of liquid is collected in an enclosed container and is released under pressure to the desired application. For modest flow rates at high pressures, rotating pumps are more appropriate. Constant volume pumps are reciprocating pumps.

Discharge pressure variations have little impact on flow rate. Any effort to restrict the amounts of energy may cause the pump case and/or discharged pipe to become over pressurized since these pumps keep supplying the same capacity. Therefore, the discharge blocker valve should never be closed when starting or using a reciprocating pump. Speed regulates the flow. There must be an automated bypass valve that is fed back to the suction supply in the few instances that call for a discharge throttle valve

A piston that oscillates back and forth through the cylinder is used in reciprocating pumps. At one end of the cylinder, two check levers for the inputs permit one-way flow, which results in a pulsed output. This form of pulp is rarely used on mobile equipment since engines' rotating movements are difficult to convert to reciprocating motions. For high-performance spectrometric pumps, two cylinders can provide a continuous output, or one cylinders can be modified to dispense with either stroke. Although the reciprocating pump discharges at such a relatively modest volume, it produces a high discharge pressure. It has a movable component that oscillates back and forth. Two ports—the inlet and outlet port are supported by gates that open and close when the cycle of operation calls for it.

Type of Reciprocating Pump:

Piston Pump

A piston pump is a type of positive displacement where the high-pressure seal reciprocates with the Piston. Piston pumps can be used to move liquid or compress gases. They can operate over a wide range of pressures. High pressure operation can be achieved without adversely affecting flow rate. Piston pumps can also deal with viscous media and media containing solid particles.[2] This pump type functions through a piston cup, oscillation mechanism where down-strokes cause pressure differentials, filling of pump chambers, where up-stroke forces the pump fluid out for use. Piston pumps are often used in scenarios requiring high, consistent pressure and in water irrigation or delivery systems

Plunger pump

Plunger pumps are pumps in which a plunger slides back and forth within a stuffing box, increasing and decreasing the working volume. Such designs are better suited for high pressures as seals are stationary within the pump, as opposed to piston designs where the seal slides within a hollow chamber.

Diaphragm pump

Diaphragm pumps are employed in almost every sector that needs to move fluids because of how adaptable they are. They are frequently employed for dewatering or removal of water in a variety of sectors. Because of their effectiveness and precision, they are utilized for filling, dispensing, and measuring.

Double acting reciprocating pump

The main operating concept is the creation of a vacuum, which results in suction, and the compression of water, which results in discharge. Each cycle is made up of two strokes. Because both strokes are effective, it is known as the Double Acting Reciprocating Pump.

Components of Reciprocating Pumps

Piston or plunger:

A piston is a component of, among other things, internal combustion engines, reciprocating pumps, gas compression, hydraulic cylinders, and pneumatic cylinders. It is the master piece confined by a cylinder and sealed by piston rings. The purpose of a piston rod and/or connect rod in an engine is to distribute force from rising gas in the barrel to the crankshaft. The role is reversed in a pump, and force is transmitted from the crank to the piston to compress or discharge the fluid inside the cylinder. In certain engines, the pistons also serves as a valve by concealing and revealing cylinder openings.

Crank and connecting rod:

The con-rod mechanism on the crankshaft converts reciprocating momentum to rotating velocity. The con-rod ties the pistons to the crankshaft, allowing combustion pressure to be transferred to the crankpin. Bearing sections are present at both ends; the piston side is referred to as the lowest part, and the flywheel side is referred to as the big end.

Suction pipe:

Suction pipes are commonly utilized in agricultural machinery for freshwater and saltwater suction and delivery. They are also utilized for irrigation and fertilizing.

Suction and delivery values:

A tube connects a soft and hard thin plastic cup to a suction hose. The cup is securely fastened to your upper chest. During a contraction, the gynecologist or midwife gently strains to assist in the delivery of your baby.

Many reciprocating pump systems have issues that might result in high maintenance costs and inconsistent operation. Vibration and noise in the pipework and the pump are two common examples. Vibration may cause valves, crossheads, crankshafts, pipework, and even pump barrels to fail and lose function. So when pulsation energy from the motor interacts the with inherent acoustic frequencies of the pipe, high amounts of pulsation might develop. For multiples of the pump speed, a reciprocating pump creates pulsation, and the amplified pulsation in the system is often worse at multiples of the plunger speed. Because most systems have had more than one resonant period, issues might arise at varying pump speeds.

Centrifugal pumps without an internally and externally self-priming stage must first be primed with both the fluid before they can begin to pump it. In furthermore, a suction-side swing safety valve or an inlet and outlet valve must be installed to prevent any syphon action and guarantee that the fluid stays in the casing once the pump has been interrupted. These impellers are stronger but slower because they are made to move moisture, which is much denser than air. The fluid being pumped and any air bubbles that are entrained are pumped into in the filtration unit in self-priming compressors that have one.

Pelton turbine and its Application

Pelton Turbine

A tangential flow impetus turbine called a Pelton Turbine uses the kinetic energy of water to create a high-speed water jet that strikes a wheel tangentially to cause it to revolve. It also goes by the name Pelton Wheel. A Pelton turbine is a type of hydroelectric turbine used in power generating. Pelton turbines, unlike the Francis and Kaplan turbines, are impulses type turbines. Although Pelton turbines are exceedingly efficient, they are not as widespread as Kaplan and Francis generators because they require a very big head of water difference the elevation between both the top and bottom reservoirs to function, limiting their application to certain geographical locations.

Figure 7. 1 Pelton Turbine

Nozzles guide a number of spoon-shaped buckets, often referred to as impulse blades, positioned around the outside rim of a drive wheel, against the powerful, high-speed jets of water (also called a runner). The direction of the water's velocity changes as it strikes the blades, following their shapes. The water seems to do a "U-turn" and emerges at the outside edges of the buckets, slowed significantly to a lower speeds. The water jet's impulse energy spins the wheel as it exerts stress on the bucket and wheel system. The wheel and subsequently a turbine get the momentum from the water jet during the process. As a result, "impulse" energy operates the turbine.

When utilized to generate power, a water reservoir is normally situated some distance well above Pelton turbine. The water is then sent via the penstock to specialized nozzles, which deliver pressured water into the turbine. The penstock is equipped with a surge tank, which absorbs unexpected variations in water that may affect the pressure. Unlike other types of turbines, which seem to be reaction turbines, the Pelton turbine is an impulse turbine. This simply implies that, rather than moving as a consequence of a counter force, water generates an impulse on the rotor to cause it to move. Figure 7. 1 Pelton Turbine

Nozzle and Flow Regulating Arrangement

Through a penstock, which has a nozzle at one end, the water from the source is transported. This nozzle may be used to create a high-speed water jet. Inside the nozzle is a moveable needle spear that is used to regulate the water flow. The Pelton Turbine's Nozzle and Flow Configuration The spear will travel axially backward and forward. The flow will decrease or stop when it is carried forward, and it will rise when this is moved backward.

Runner and Buckets

A Pelton turbine is made up of a runner, which really is a circular disc on which several buckets are attached with identical distances between them. Either double ellipsoid or double ellipsoidal-shaped buckets are installed. Pelton Wheel Turbine's Runner and Buckets each bucket has a splitter, a separating wall that divides the bucket onto two equal pieces. Depending on the head of the inlet of the Pelton turbine, the buckets are often composed of cast iron, stainless steel, or bronze.

Casing

The Casing encloses the whole system of runners and buckets, as well as the intake and brake jets. The Pelton turbine's casing does not engage in any hydraulic functions, although it does stop water from sprinkling while it is in operation and facilitate water discharge toward the tail race. The functioning of a Turbine blade is straightforward. High-speed water jets come from the cylinders that surrounding the turbine in this type of turbine. These nozzles are designed such that the water jet hits the bins at splitters, which are located in the center of the bucket and separate the water jet into two streams. The two distinct streams then run down the inner arc of the bucket, exiting in the reverse direction that they entered. The change in velocity of the water causes an urge on the turbine blades, resulting in torque and spinning in the turbine.

The high-velocity gas encroaches on the blade, converting a considerable percentage of the flowing gas stream's kinetic energy into turbine input shaft. In the nozzle, the static pressure falls as the absolute velocity increases. The rotor's absolute velocity is thus lowered, while the static pressure and relative velocity remain unchanged. To transmit the most energy, the blades must revolve at roughly half the speed of the gas jet. The impeller has a response degree of zero by definition. This level of reactivity implies that the full enthalpy decrease is received in the nozzle, as well as the exit velocity.

Water is transported by a penstock, which is a pressurized water conductor. The penstock links the upper storage to the turbine and forms water suction component of the turbine. A spray nozzle is located at the Penstock's end. The spray nozzle turns the theoretical energy of flowing water into kinetic energy, which manifests as a supersonic speeds stream of water shot out of the tip and toward the Pelton runner baskets. The water jet makes tangential contact with the interior surface of each bucket.

Each bucket is divided into two half by a high ridge known as either a splitter. The splitter separates the water jet because it flows evenly to both side of the bucket. Each bucket has a notch that allows the fluid flow to enter at the optimal angle. The bucket's spoon shape progressively converts the kinetic energy of water jet to mechanical energy as the water accomplishes a 180-degree revolution in the bucket. The mechanical energy appears as torque upon that runner shaft, causing the runner to revolve. The water is dumped through a disposal pit after exiting the bucket

Venturimeter

The rate of flow of a fluid moving through a pipe or an open channel is measured via fluid flow measurement. A venturimeter measures the rate of fluid flow via a pipe. It is a device that converts pressure energy to kinetic energy and is used to quantify the rate of flow through a pipe. A venturimeter, in other terms, is a tube with a narrowed neck that increases velocity while decreasing pressure. They are used in pipelines to measure the flow rate of compressible and incompressible flow. The venturimeter operates on the basis of Bernoulli's equation. It has a converging part, a neck, and a diverging portion (Figure 7.2).

Parts of a Venturimeter

1. Converging part
2. Throat
3. Diverging part

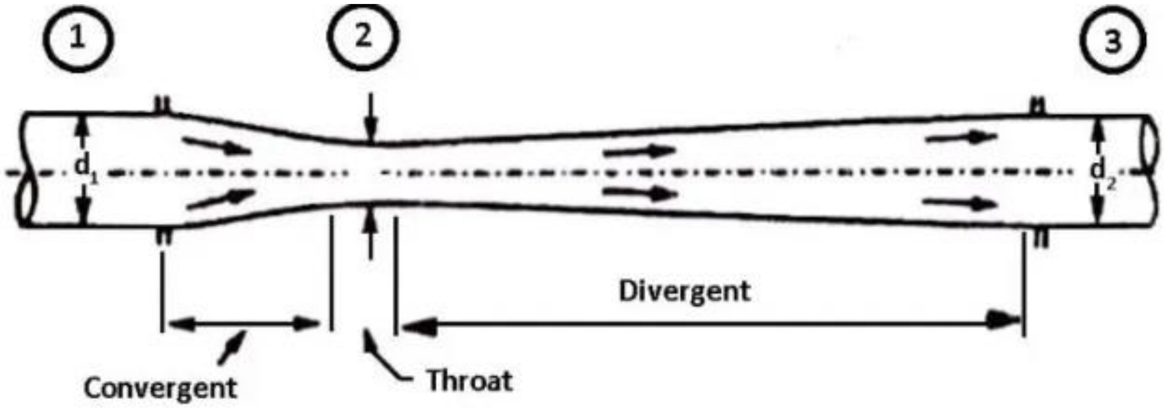

Figure 7.2 Venturimeter

Converging part

The convergent section of a venturimeter is always smaller than the divergent portion. This is done to provide a quick converging transit and a slow diverging passage in the flow direction, avoiding energy loss due to separation.

Throat

The throat is a tiny section of a pipe with a constant diameter d2.

Diverging Part

A venturimeter is made up of two conical sections separated by a short uniform cross-sectional segment. This little portion, known as the neck, has the smallest surface area.

Working of Venturimeter

The liquid is accelerated as it passes through the venturimeter and the convergent cone. As a consequence of the compression, the velocity of the liquid as in throat increases more than it did at the convergent portion. This increase in velocity reduces the strain on the neck significantly. If the pressure head at the neck falls below the divergence head (which is 2.5 metres of water), the liquid flow will tend to separate. This is known as the venturi vacuum.

To eliminate the propensity of separation there at throat, there is always a predetermined ratio of the neck and pipe diameters (i.e., d2/d1). The proportion should be 1/3 to 1/2. The diffuser in a venturi metre ensures a slow and constant deceleration beyond the throat. It is intended to guarantee that the pressure returns to near-original levels before the venturimeter. If the pressure quickly recovers in the divergent region, the liquid stream has a good chance of breaking free of the walls. To prevent this, the diverging cone is lengthened. Another rationale for lengthening the diverging cone is to decrease frictional losses. As a result, it is three to four times longer than convergent cone.

Discharge Though Venturimeter

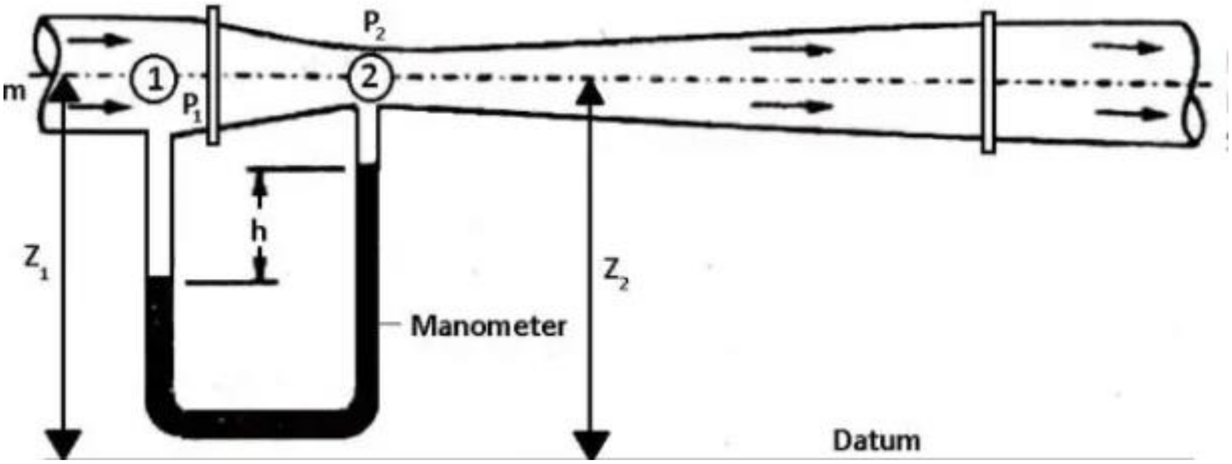

Figure 7.3 Discharge Though Venturimeter

According to NPTEL, the converging portion's job is to increase the fluid's velocity while briefly lowering its static pressure. There is a pressure differential between the intake and the throat. This pressure differential is proportional to the rate of low.

The theoretical flow rate is calculated by using the formula and the energy equation at the input and throat sections, and assuming the fluid is ideal, as follows: Figure 7.3 Discharge Though Venturimeter

Let,

- p1 = Pressure at section 1,
- v1 = Velocity of water at section 1,
- z1 = Datum head at section 1,
- a1 = Area of venturi meter at section 1, and
- p2, v2, z2, a2 = are corresponding values at section 2
- C = Venturi meter

$$Venturi\ head = \left[\frac{13.6 - \omega}{\omega}\right] \times Head\ of\ mercury$$

Where,

13.6 = Specific gravity of mercury, and

ω = Specific weight of the oil

By applying Bernoulli’s equation at sections (1) and (2), we can get

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$

As the pipe is horizontal, hence z1 = z2

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} = \frac{p_2}{\rho g} + \frac{v_2^2}{2g}$$

Or

$$\frac{p_1 - p_2}{\rho g} = \frac{v_2^2}{2g} - \frac{v_1^2}{2g}$$

But p1-p2/ρg is the difference of pressure heads at sections 1 and 2 and it is equal to h or p1-p2/ρg = h

Substituting this value of p1-p2/ρg in the above equation, we get

$$h = \frac{v_2^2}{2g} - \frac{v_1^2}{2g}$$

Now, we need to apply the continuity equation in sections 1 and 2,

$$a_1 v_1 = a_2 v_2 \quad Or \quad v_1 = \frac{a_2 v_2}{a_1}$$

Substituting this value of v1 in the equation,

$$v_2 = \sqrt{2gh \frac{a_1^2}{a_1^2 - a_2^2}} = \frac{a_1}{\sqrt{a_1^2 - a_2^2}} \sqrt{2gh}$$

$$\iota_2 \frac{a_1}{\sqrt{a_1^2 - a_2^2}} \times \sqrt{2gh} = \frac{a_1 a_2}{\sqrt{a_1^2 - a_2^2}} \times \sqrt{2gh}$$

Gives the discharge under ideal conditions and is called theoretical discharge. Therefore, the actual discharge will be less than the theoretical discharge (Figure 7.4)

$$Q_{act} = C_d \times \frac{a_1 a_2}{\sqrt{a_1^2 - a_2^2}} \times \sqrt{2gh}$$

Types of Venturimeter

Horizontal Venturimeter

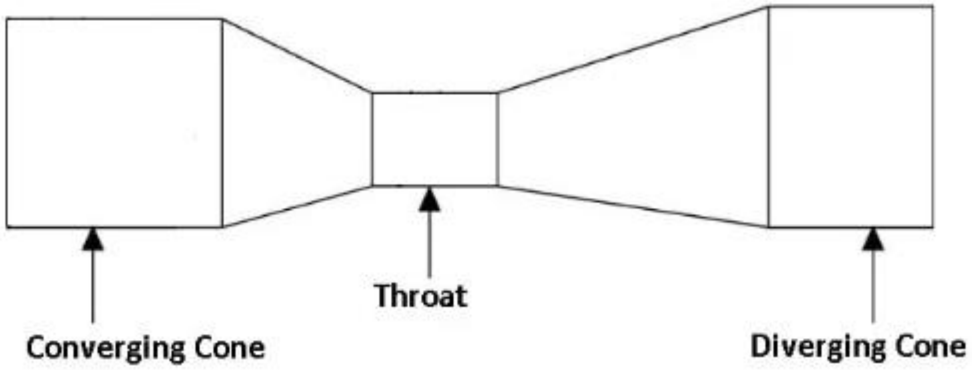

Figure 7.4 Horizontal Venturimeter

Horizontal venturimeters have the most kinetic energy and the least potential energy. Water flow is measured using a horizontal venturimeter with an input diameter of 200 mm and a throat diameter of 100 mm.

Veritcal Venturimeter

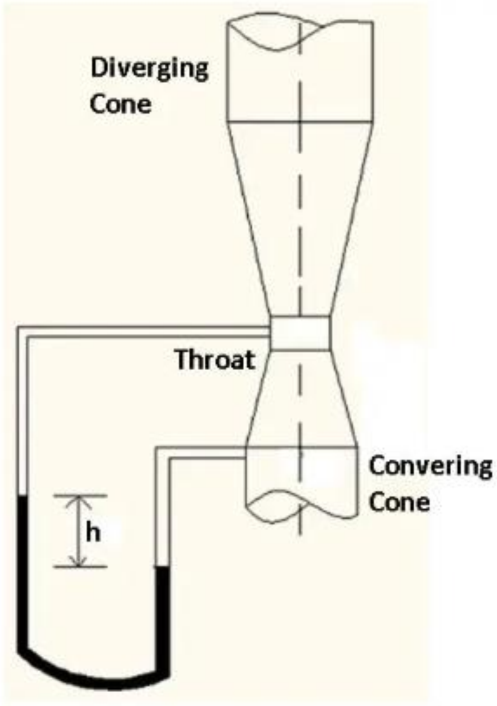

Figure 7.5 Veritcal Venturimeter

Vertical venturimeters have the most potential energy and the least kinetic energy. These are typically equipped with a circular pipe 30 cm in diameter and a throat diameter of 15 cm. The readings of the two portions of the manometer varied by 30 cm. You can quickly determine the volume of water that passes through the pipe with this kind (Figure 7.5).

Inclined Venturimeter

Vertical venturimeters have the most potential energy and the least kinetic energy. These are typically equipped with a pipe 30 cm in diameter and a throat diameter of 15 cm. The readings of both the two portions of the manometer varied by 30 cm. You can quickly determine the volume of water that passes through the pipe with this kind. A Both the kinetic and potential energy of this kind are intermediate between the two.

An inclined venturimeter is typically installed in a vertical plane into an inclined pipe to measure the flow through to the pipe. The key to good functioning is to instal an appropriate venturi metre. As a result, venturimeters should be placed in accordance with the manufacturer's instructions. When installing venturi metres, the following rules should be followed:

1. The flow direction in the venturi metre should be examined and set up in order to agree on the flow direction.

2. When fitting the flanges on the venturimeter, ensure that the pipe flanges are correctly aligned with the venturimeter ends.

3. Remember that pipe stabilizers should not be used on venturimeters.

4. When securing the prerequisite for successful with a bolt, take care not to overtighten it.

5. Tolerances should be applied in accordance with industry norms.

Horizontal orientation is required for pressure taps used in liquid service applications.

Advantages

1. When compared to other kinds of fluid measuring equipment, the power loss is deemed negligible.

2. These are used in situations when a tiny head is available.

3. Over a large flow range, accuracy will improve.

4. The co-efficient of discharge in a venturimeter is very high.

5. These devices are simple to operate and can handle both compressive and incompressible fluids.

6. Venturimeters are dominated by high flow or discharge rates.

Disadvantages

1. Venturimeters are not cheap to instal.

2. These gadgets must be maintained.

3. This method takes up more room than the orifice metre.

4. The venturimeter is more expensive and larger.

5. Where the wall thickness is 76.2 mm, they are not utilised.

Applications

1.The flow rate of the fluid may be simply determined with a venturimeter.

2. It is used in industry to calculate the pressure of such a volume of gas or liquid within a pipe.

3. These are very useful for measuring airflow in automotive carburetors.

4. It is used in process industries to measure and regulate process flow.

5. They are also used in the medical profession to measure arterial blood flow using a venturimeter.

6. This equipment is used in the treatment of wastewater.

7. It is also handy when high-pressure recovery is required.

Hydraulic Machines

Hydraulic machines typically employ a liquid fluid as the power source to do tasks. . Vehicle construction is a typical example. The hydraulic machine pumps hydraulic fluid to different hydraulic motors and even hydraulic cylinders throughout the machine, and it then gets pressured according to the resistance present. The fluid we just saw may be managed directly or automatically by regulating valves and transferring it via hoses, tubes, or pipes.

The hydraulic system is similar to pneumatic systems in that they are founded on Pascal's law, which asserts that any pressure applied to a fluid within a closed system would transfer that pressure equally evenly and in all directions. A hydraulic system, for example, employs an inviscid liquid as its fluid rather than a compressible gas.

Applications of Hydraulic Machines

Hydraulic Lifts: A hydraulic lift is an elevator that is powered by fluid pressure created by a suitable fluid. It is often used to lift vehicles in service stations and even garages. A hydraulic lift consists of two pistons separated by a liquid-filled gap. A piston with a tiny cross-section A1 is employed to exert a force on the liquid that is, say, F1. The pressure stated by P = F/A is passed via the liquid to the bigger cylinder, which is coupled to a larger piston of area defined as A2, resulting in an upward force given by P A2. As a result, we may state that the crankshaft is capable of bearing a considerable force, such as the weight of a vehicle or truck put on the platform. The platform may be moved down and up by altering the force at A1. As a result, we can say that the applied force has been enhanced by a factor of A2/A1, where A2/A1 is the mechanical advantage of the device.

Hydraulic Breakers

Hydraulic breakers are a braking mechanism arrangement in which appropriate brake fluid is employed to transmit pressure from the control mechanism to the brake mechanism. Automobile hydraulic brakes operate on the same basis. When we apply a little amount of effort to the pedal with our foot, the master piston moves within the master cylinder, and the pressure created is communicated via the brake fluid, which acts on a bigger piston. A substantial force usually exerted on the piston, pushing it down and extending the brake shoes against the brake lining. As a result, we can observe that a modest effort on the pedal results in a huge retarding force on the wheels.

One significant benefit of the hydraulic brake system is that the pressure created by pushing a pedal is sent evenly to all cylinders that are normally coupled to the four wheels, resulting in equal braking effort on all four wheels.

Applications of Hydraulic Machines

The use of hydraulics in daily life has grown so common that no one ever thinks about it. Today, as we all know, hydraulics are used to move machinery and equipment to fulfil numerous jobs, including tractors in agriculture. Cranes in building and construction, forklifts in manufacturing and production warehouses, brakes in transportation, and many more are examples. Hydraulic machines generally use hydraulic fluid pressure to propel movement or as a primary source of eergy. Dump trucks, aluminium extruders, plastic extruders, cranes, jackhammers, and hose crimpers are further examples of hydraulic equipment.

Hydraulics in Daily Life

Metal stamping and hose crimping, injection moulding, and other operations done by hydraulic machinery may also be included.

Spinning motors are a tremendous source of enjoyment at amusement parks, as we can see below. They use hydraulics technology to power rides that then produce motion, such as a Ferris wheel.

Vehicle Braking - Hydraulics power almost all vehicles on the road. Take, for example, brake fluid, which is a vital component of a vehicle's braking system. The pressing of the brake pedal causes a rod and piston inside the master cylinder to move and create the intended effect, which ideally entails slowing or halting the vehicle.

Lifting and Repair & Maintenance - As we all know, it would be quite impossible to raise a very large motor vehicle for repair and maintenance without a hydraulic system. The device typically employs hydraulic fluid to assist in lifting any heavy weight to the appropriate height.

CHAPTER 8

BOUNDARY LAYER SEPARATION

Ramachandran T,
Professor, Department of Mechanical Engineering, Jain (Deemed-to-be University) Bangalore, India
Email Id-t.ramachandran@jainuniversity.ac.in

Sir George Cayley conceived and manufactured the first modern aerofoil, which was subsequently employed on a hand-launched glider, at the beginning of the nineteenth century, after examining the highly cambered narrow wings of many different birds. This biomimetic, strongly cambered, and thin-walled design remained the dominant aerofoil form for about 100 years, owing to the fact that the real mechanics of lift and drag were not scientifically known but were examined empirically. One of the primary issues with these early aerofoil designs was that they suffered boundary layer separation at relatively low angles of attack. This drastically restricted the amount of lift that the wings could generate, requiring larger and bigger wings to allow for any development in terms of aircraft size. Despite the lack of analytical methods to investigate this issue, aerodynamicists continued to favour tiny aerofoil sections, citing enough evidence in nature to support their usefulness. The difficulty was thought to be one of degree, i.e. progressively iterating natural aerofoil designs, rather than type, i.e. inventing an altogether new aerofoil shape in accordance with basic physics.

During the pre-WWII period, designers' faulty ideas were exacerbated by the increased use of wind tunnel testing. The wind tunnels utilised at the time were modest and operated at extremely low flow rates. This meant that the aerofoils' performance was being evaluated in laminar flow (smooth flow in layers, no mixing perpendicular to flow direction) rather than turbulent flow (mixing of flow through tiny vortices) across the wing surfaces. Under laminar flow circumstances, increasing an aerofoil's thickness increases the amount of skin-friction drag (as shown in last month's piece), hence thinner aerofoils were thought to be preferable.

In 1915, the modern aeroplane was created.

During WWI, Germany's condition altered considerably. Hugo Junkers invented the first practical all-metal aeroplane with a cantilevered wing in 1915, which was basically the same semi-monocoque wing box design that is being used today. The most common design at the time was a biplane held together by wires and struts, which created significant quantities of parasitic drag and so restricted the maximum speed of the aircraft. Because these supporting struts and cables were removed, the flight loads had to be borne by other ways. To counteract the up and down bending stresses created by lift, Junkers cantilevered a beam from each side of the fuselage, the main spar, at roughly 25% of the chord of the wing. Then, at 75% of the chord, he installed a smaller second spar known as the trailing edge spar to aid the main spar in resisting fore and aft bending caused by wing drag. The external wing skin linked the two spars to form a closed box-section known as the wing box. Finally, the "D"-shaped leading edge was formed by fitting a curved piece of metal to the front of the wing, and the trailing edge was formed by running two pieces of metal out. This sequence of three closed sections gave the wing with enough torsional stiffness to withstand the twisting loads caused by the offset of the centre of pressure (the point at which the lift force may be regarded to operate) from the shear centre (the point where a vertical load will only cause bending and no twisting). Junker's innovations were all merged in the world's first practical all-metal aeroplane, the Junker J 1, which, while being much heavier than other aircraft at the time, became the dominant method of construction for the bigger and faster aircraft of the next generation.

Because of the space necessary for internal bracing, Junkers construction naturally resulted in a substantially thicker wing, and this design supplied the motivation for revolutionary aerodynamics research. Ludwig Prandtl, who conducted his famed aerodynamics research at the University of Göttingen, backed Junker's theories. Prandtl earlier established the concept of the boundary layer, which is defined as the presence of a U-shaped velocity profile with a no-flow condition at the surface and a rising velocity field towards the main stream some distance away from the surface, as mentioned in last month's article. The existence of a boundary layer, according to Prandtl, supported the simplistic premise that fluid flow may be divided into two non-interacting portions: a thin layer near to the surface regulated by viscosity (the stickiness of the fluid) and an inviscid mainstream. Prandtl and his colleagues were able to generate far more accurate forecasts of the lift and drag performance of certain wing-shapes, which substantially aided in the design of German WWI aircraft. Prandtl demonstrated in 1917 that Junker's thicker and less-cambered aerofoil section provided much better lift characteristics than the typical narrower portions utilised by Germany's adversaries. Second, the broad aerofoil allowed the aircraft to fly at a considerably greater angle of attack without stalling, improving the plane's manoeuvrability while dog fighting.

Pressure Drag vs. Skin Friction

A boundary layer's flow may be either laminar or turbulent. Laminar flow is ordered and stratified, with no fluid particle interaction between individual layers, while turbulent flow has extensive fluid exchange perpendicular to the flow direction. The physics of the boundary layer are heavily influenced by the kind of flow. A turbulent boundary layer, for example, is thicker than a laminar one owing to the higher area of mass exchange and also has a steeper velocity gradient near to the surface, i.e. the flow speed rises more rapidly as we go away from the wall.

Laminar versus turbulent boundary layer velocity profiles

The velocity profile of a laminar boundary layer vs a turbulent boundary layer. Take note of how the turbulent flow accelerates velocity away from the wall.

Layers of fluid in the boundary layer, like your hand moving over a surface, suffer friction, i.e. the slower areas of the flow are holding back the faster sections. This indicates that the velocity differential across the boundary layer causes internal shear stresses, similar to friction acting on a surface. This sort of friction is known as skin-friction drag, and it is most common in streamlined flows when the bulk of the body's surface is parallel to the flow. A streamlined body suffers increased drag when the boundary layer flow over its surfaces is turbulent because the velocity gradient at the surface is bigger for turbulent than for laminar flow. An aircraft wing in cruise is a classic example of a streamlined body, thus it's no surprise that sustaining laminar flow across aircraft wings is a hot study area.

We can disregard pressure fluctuations in the flow direction over flat surfaces. The boundary layer stays stable under these circumstances, although it thickens in the flow direction. Of course, this is an idealised situation, and in real-world applications such as curved wings, the flow is most likely encountering an unfavourable pressure gradient, i.e. pressure rises in the flow direction. The boundary layer may become unstable and detach from the surface under certain circumstances. The separation of the boundary layer causes a second sort of drag known as pressure drag. This form of drag is most common in non-streamlined bodies, such as a golf ball travelling through the air or an aeroplane wing with a high angle of attack.

Consider fluid flow across a cylinder to address this topic. Fluid particles must come to rest just at the front of the cylinder. This is the point of greatest pressure and is appropriately named the stagnation point (to conserve energy the pressure needs to fall as fluid velocity increases, and vice versa). Further downstream, the flow lines curve due to the curvature of the cylinder, and in order to equilibrate the centripetal forces, the flow accelerates and the fluid pressure lowers. As a result, a region of accelerating flow and lowering pressure develops between the stagnation point and the cylinder's poles. Because of all the free space downstream of the cylinder, the curvature of the cylinder is less efficient in directing the flow in curved streamlines as the flow reaches the poles. As a result, the flow curvature decreases and the flow slows, transforming the previously beneficial pressure gradient into an unfavourable pressure gradient with increasing pressure.

Separation of boundary layers across a cylinder

To comprehend boundary layer separation, we must first comprehend how these positive and negative pressure gradients impact the form of the boundary layer. We know from our consideration of boundary layers that the fluid moves slower as we go closer to the surface owing to the retarding influence of the no-slip condition at the wall. The lowering pressure along the streamlines serves to push the fluid forward in a favourable pressure gradient, offsetting some of the decelerating effects of the fluid's viscosity. As a consequence, the fluid is not decelerated as much near to the wall, resulting in a broader U-shaped velocity profile and slower growth of the boundary layer.

By example, when there is an unfavourable pressure gradient, the mainstream pressure rises in the flow direction, slowing the flow in the boundary layer. As a result, when there is an unfavourable pressure gradient, the pressure forces strengthen the retarding viscous friction forces at the surface. As a consequence, the difference in flow velocity between the wall and the mainstream becomes more obvious, and the boundary layer expands faster. If the unfavourable pressure gradient operates over a sufficiently long distance, the flow slowdown is sufficient to reverse the flow direction in the boundary layer. As a result, the boundary layer produces a point of inflection known as the point of boundary layer separation, beyond which a circular flow pattern forms.

When it comes to aircraft wings, boundary layer separation may have serious effects ranging from increased pressure drag to a sudden loss of lift, known as aerodynamic stall. An aeroplane wing is just an extended and perhaps asymmetric version of the cylinder seen above. As a result, airflow across a wing's upper convex surface follows the same fundamental principles mentioned above: At the cutting edge, there is a point of no return.

Up to the point of maximal thickness, an area of increasing mainstream flow (favourable pressure gradient). Beyond the point of maximum thickness, an area of decelerating mainline flow (adverse pressure gradient). Boundary layer separation is a critical problem for aircraft wings because it creates a huge wake that affects the flow downstream of the site of separation. Skin-friction drag occurs as a result of the fluid's intrinsic viscosity, i.e. the fluid adheres to the surface of the wing and the accompanying frictional shear stress imposes a drag force. When a boundary layer splits, a drag force is created due to pressure differences upstream and downstream of the wing. The entire size of the wake, and hence the degree of pressure drag, is affected by the point of separation along the wing. The velocity profiles of turbulent and laminar boundary layers (shown above) reveal that the fluid velocity grows significantly slower away from the wall in a laminar boundary layer. As a consequence, flow in a laminar boundary layer will reverse direction significantly sooner than flow in a turbulent boundary layer in the presence of an unfavourable pressure gradient

To summarise, we now know that a fluid's intrinsic viscosity causes the existence of a boundary layer with two alternative sources of drag. Skin drag is caused by frictional shear stress between the fluid and the surface, whereas pressure drag is caused by flow separation and the presence of a downstream wake. Because total drag is the combination of these two effects, the aerodynamicist is forced to make a difficult decision: Laminar flow reduces skin friction drag by lowering shear stress at the wall, but this raises pressure drag as the boundary layer separates.] Turbulent flow reduces pressure drag by delaying boundary layer separation, but this increases skin friction drag owing to increased shear stresses at the wall.

As a consequence, neither laminar nor turbulent flow can be considered desirable in general, and a decision must be made based on the individual application. A turbulent boundary layer is ideal for a blunt body, such as a cylinder, since pressure drag prevails. For more streamlined bodies, such as an aeroplane wing in cruise, skin-friction drag dominates the total drag, making a laminar boundary layer ideal. Dolphins, for example, have highly streamlined bodies that allow them to maintain laminar flow. Early players, on the other hand, discovered that worn rubber golf balls flew further than perfect ones, which led to the development of dimples on golf balls. Because of the low flight speeds, fluid flow over golf balls is mostly laminar. Dimples are therefore nothing more than little flaws that convert the primarily laminar flow into a turbulent one, delaying the commencement of boundary layer separation and so reducing pressure drag.

Stall in Aerodynamics

Aerodynamic stall is the second and more dramatic impact of boundary layer separation in aircraft wings. The unfavourable pressure gradient acting on the top surface of the wing is benign at relatively low angles of attack, such as during cruise, and the boundary layer stays attached throughout the whole surface. However, when the angle of attack increases, so does the pressure gradient. The boundary layer will begin to split along the trailing edge of the wing at some point, and this separation point will move upstream as the angle of attack increases. Separation will occur extremely near to the point of maximum thickness of an aerofoil if it is positioned at a sufficiently enough angle of attack, and a huge wake will emerge behind the point of separation. This wake redistributes the flow throughout the remainder of the aerofoil, reducing the lift created by the wing substantially. As a consequence, the lift generated is significantly decreased, resulting in an aerodynamic stall. Because of the high pressure drag created by the wake, the aircraft might lose even more velocity, pushing the separation point farther upstream and producing a negative feedback loop in which the aircraft physically begins to fall out of the sky in an uncontrolled spiral. To avoid complete loss of control, the pilot must reconnect the boundary as soon as possible, which is accomplished by lowering the angle of attack and pointing the aircraft's nose down to gain speed.

The lift generated by a wing is provided by

L = \frac{1}{2}

C L \rho V2 S, where rho is the density of the surrounding air, V is the flying velocity, S is the wing area, and C L is the aerofoil shape's lift coefficient. The lift coefficient of a certain aerofoil design grows linearly with angle of attack until it reaches a maximum point C Lmax. A conventional aerofoil's maximum lift coefficient is approximately 1.4 at an angle of attack of roughly 16circ, which is constrained by the critical angle of attack when the stall condition occurs.

During cruise, the angle of attack is very modest (about 2circ) because the high flight velocity V ensures adequate lift. Furthermore, we want to maintain a narrow angle of attack since it reduces the pressure drag caused by boundary layer separation. However, the flying velocity is significantly lower during takeoff and landing, requiring the lift coefficient to be raised by putting the wings at a more aggressive angle of attack (about 15circ). The problem is that even with a near maximum lift coefficient of 1.4, huge jumbo planes struggle to provide the requisite lift power at safe landing speeds. While it is conceivable to expand the wing area, this would have a negative impact on the aircraft weight and hence fuel economy.

Devices for lifting heavy objects

Leading-edge slats and trailing-edge flaps are a considerably more elegant approach. A slat is a narrow, curved aerofoil attached to the front of a wing to create secondary airflow via the space between the slat and the leading edge. The air accelerates across this gap, injecting high momentum fluid into the top surface boundary layer and delaying the beginning of flow reversal. Similarly, one or two curved aerofoils may be positioned towards the trailing edge of the wing to energise the flow. The high momentum fluid reinvigorates the flow that has been halted by the unfavourable pressure gradient in this situation. These devices often quadruple the maximum lift coefficient, allowing large jumbo planes to land and depart at relatively low runway speeds.

Buckingham's Pi theorem

Any equation that relates dimensionless products is obviously dimensionally homogenous; that is, the form of the equation does not rely on the basic units of measurement. This remark may be expressed officially as follows:

Being reducible to an equation among dimensionless products is a necessary requirement for an equation to be dimensionally homogenous. E. Buckingham deduced the essential principle that the criteria of this theorem must also be met. Buckingham's theorem is so phrased as follows:

'If an equation is dimensionally homogenous, it may be reduced to a connection among a whole set of dimensionless products,' says one researcher. "Any equation that relates a dimensionless product is dimensionally homogenous," in other words. The criterion that an equation be dimensionally homogenous is applied to an equation among dimensionless products".

This theorem is far from self-evident. Buckingham did not formally prove the theorem, but he did give evidence that made its correctness seem feasible. Buckingham's theorem encompasses the whole dimensional analysis theory. However, before this theorem was developed, dimensional analysis concepts were used.

Consider the situation in Section 3.1 in light of Buckingham's theorem. Only that the five variables are connected by a dimensionally homogeneous equation are we assuming. This may be represented as f(F,V,D,,) = 0, where f is an unnamed function.

According to the Buckingham's theorem, if there are n dimensional variables involved in a phenomenon that can be completely described by m fundamental quantities or dimensions (such as mass, length, time, etc.) and are related by a dimensionally homogeneous equation, then the relationship between the n quantities can always be in terms of of exactly (n-m) dimensionless and independent p terms.

Procedure for Buckingham's Pi method

Mathematically, if any variable Q1 depends on the independent variables Q2, Q3, Q4……Qn, ; the fundamental equation may be written as,

$$Q1 = f\,(Q2, Q3, Q4 \ldots\ldots Qn)$$

Which can be transformed to another functional relationship as,

$$f1\ (Q1, Q2, Q3, Q4 \ldots\ldots Qn) = C$$

Where 'C" is the dimensionless constant.

In accordance with the π theorem, a non-dimensional equation can thus be obtained in the form,

$$f2\ (\pi 1, \pi 2, \pi 3, \pi\ n\text{-}m) \quad = \quad C1$$

Wherein, each dimensionless π-term is formed by combining m variables out of the total n variables with one of the remaining (n-m) variables. These 'm' variables which appear repeatedly in each of the π terms, are called repeating variables. These are 'm' fundamentals quantities. They themselves do not form a dimensionless parameter. Thus the different π terms may be established as,

$$\pi 1 \quad = \quad Q1a1, Q2b1, Q3c1 \ldots.. Qmm1\ Qm{+}1$$

$$\pi 2 \quad = \quad Q1a2, Q2b2, Q3c2 \ldots.. Qmm3\ Qm{+}2$$

$$\pi n\text{-}m \quad = \quad Q1an\text{-}m, Q2b\ n\text{-}m, Q3c\ n\text{-}m, \ldots.. Qmm\ n\text{-}m, Qn$$

In the above equation, each individual equation is dimensionless and the exponents a, b, c, d …….. m etc are determined by considering dimensional homogeneity for each equation such away that each π term is dimensionless. The final general equation for the phenomenon may then be obtained by expressing any one of the π terms as a function of the others.

$$\pi 1 \quad = \quad f1\ (\pi 1, \pi 2, \pi 3, \ldots\ldots.. \pi\ n\text{-}m$$

$$\pi 2 \quad = \quad f2\ (\pi 1, \pi 2, \pi 3, \ldots\ldots.. \pi\ n\text{-}m$$

Consider the problem of drag force that a smooth spherical body experiences in a stream of compressible fluid. Assume that five variables namely, drag force (F), velocity (V), diameter of sphere (D), density of fluid (ρ) and absolute viscosity (μ), are related by a dimensionally homogeneous equation. This may be indicated by

$F = f(V, D, \rho, \mu)$.

This may be written in the general form

$f1\ (F, V, D, \rho, \mu) = C$

The total number of variables, $n = 5$

These variables may be completely described by three fundamental dimensions of M – L – T (m = 3 fundamental unit).

Therefore the number of π terms = $n - m = 5 - 3 = 2$ (ie., Number of π terms is 2)

$$f2\ (\pi 1, \pi 2) \quad = \quad C1$$

In order to form these π terms, choose the three repeating variables following the guidelines, since the fundamental dimensions are three. Choose r, V and D as repeating variables. Since physical quantities of the similar dimensions can neither be added nor subtracted the terms are expressed as products as follows:

$$\pi1 = ra1Vb1Dc1m$$

$$\pi2 = ra2Vb2Dc2F$$

Expressing π1 dimensionally in terms of M- L - T, we get

Equating the exponents of M, L, and T, we get

For M: $a1 + 1 = 0; a1 = -1$

For L: $-3 a1 + b1 + c1 - 1 = 0$

For T: $-b1 - 1 = 0; \quad b1 = -1$

Buckingham's theorem says that, since the equation is dimensionally homogenous, f is a function of a comprehensive series of dimensionless products of the variables, which consists of the pressure coefficient, P = F-1V-2D-2, and the Reynolds number, R = VD-1. As a result of Buckingham's theorem, the equation may be reduced to the form f (P,R) = 0. This connection may be expressed explicitly as P = f. (R). If f is only a symbol for any function, the relationships f (P, R) = 0 and P = f(R) imply the same thing: it is feasible to create a curve that shows the relationship ship between P and R. The equation, P = f(R), yields the same result as Rayleigh's technique above. The logic that lead to the result, P = f(R), is not limited to spherical bodies; it applies to any form of body. Consider an aircraft wing. The geometry of the curve who connects P and R is, of course, determined by the shape of the body. Dimensional analysis yields no information on the shape of the curve.

Rayleigh's approach of dimensional analysis is not fundamentally different from Buckingham's method. The algebraic stages in both systems are almost identical. Buckingham's technique, on the other hand, frees us from the promiscuous use of infinite series. Too frequently, the development of an infinite series is not described as a logically necessary step in Rayleigh's technique. As a result, the impression is formed that the standard deviation in a physical issue may be arbitrarily equated to something like a product of the independent variables' powers and a numerical coefficient.

This assumption is occasionally a valid approximation, especially in heat transfer issues, although it is not required for dimensional analysis.

Buckingham's theorem allows us to conclude that if n variables are connected by an unknown dimensionally homogeneous equation, the measurement can be expressed in the form of a relationship among (n - m) dimensionless products, where (n - m) is the number of goods in a complex set of dimensionless products of the variables. Most of the time, m equals the number of basic dimensions in the issue. However, this is not an ironclad rule since the number of basic dimensions in an issue might vary depending on the fundamental dimension system utilised.

Turbine

A turbine is a device that converts the kinetic energy of a fluid, such as water, steam, air, or combustion gases, into rotational motion of the device. Turbines are often found in power generating, engines, etc propulsion systems. Turbines (particularly turbomachines) are machines because they transfer and change energy. A basic turbine is made up of a number of blades - nowadays, steel is one of the most often used materials - that enable fluid to access the turbine and drive the blades. While the fluid runs through, these blades rotate, trapping part of the energy as rotary motion. Fluid passing through a turbine lost kinetic energy and departs with a little less energy than it entered.

Turbines are employed in a variety of applications, and each sort of turbine requires a somewhat different architecture to do its function well. Wind turbines, hydro turbines, heat engines, and propulsion all require turbines. Turbines are critical because practically all power is generated by converting mechanical power from a turbine into electricity through a generator. A turbine is any of many machines that transform the energy in a fluid stream into mechanical energy. The fluid is often converted by passing it through a set of fixed channels or vanes that alternate with passages made up of finlike blades connected to a rotor. The rotor revolves and work is extracted by organising the flow such that a closed loop, or torque, is applied on the rotor blades. Turbines are grouped into four categories based on the fluids they use: water, steam, gas, and wind. Although all turbines operate on the same principles, their unique designs vary enough to warrant distinct descriptions A water turbine converts the potential energy originating from the elevation difference between an upstream water container and the turbine-exit water level (the tailrace) into work. Water turbines are the contemporary descendants of basic waterwheels, which have been around for around 2,000 years. Water turbines are now mostly used to generate electricity.

However, steam turbines connected to electric generators provide the most electrical energy. The turbines are powered by steam generated by either a fossil-fuel or nuclear-powered generator. The energy extracted from steam may be readily stated in terms of the enthalpy change throughout the turbine. The total of internal thermal energy and the product of pressure time’s volume yields enthalpy, which represents both thermal and mechanical energy forms in a flow process. The available enthalpy change via a steam turbine rises with steam generator temperature and pressure, as well as with decreasing turbine-exit pressure.

The energy taken from the fluid in a gas turbine may also be described in terms of the enthalpy change, which is closely proportional to the temperature decrease throughout the turbine. The working fluid of gas turbines is air combined with gaseous combustion products. A compressor, a combustion chamber, and a turbine are all standard components of gas turbine engines. These are often placed as an entire unit and function as a full prime mover on an open cycle, in which air is sucked out of the atmosphere and combustion products are subsequently released back into the environment. Because proper functioning is dependent on the interaction of all components, the whole device, which is essentially an internal-combustion engine, must be considered rather than the turbine alone. Wind energy may be captured using a wind turbine to generate electricity or to pump water from wells. Wind turbines are the descendants of windmills, which were significant sources of energy from late middle Ages through the nineteenth century.

Water turbines

Water turbines are commonly classified into two types: (1) impulse turbines, which are used for high water heads and low flow rates, and (2) reaction turbines, which are used for low water heads and medium or high flow rates. The Pelton impulse turbine and reaction turbines of the Francis, propeller, Kaplan, and Deriaz varieties are included in these two classifications. Turbines may be configured with either horizontal or vertical shafts. Within each kind, wide design modifications are feasible to satisfy the individual local hydraulic circumstances. Most hydraulic turbines are now utilised to generate power in hydroelectric plants.

Turbines on impulse

The potential energy, or the head of water, is first turned into kinetic energy in an impulse turbine by discharging water via a precisely constructed nozzle. The released jet is directed onto curved buckets mounted to the runner's edge to absorb water energy and convert it to productive world. Modern impulse turbines are based on a concept invented in 1889 by Lester Allen Pelton, an American engineer. Tangentially, the free water jet impacts the turbine buckets. Each bucket has a high centre ridge to split the flow and allow the runner to exit on both sides. Pelton wheels are best suited for high heads, often over 450 metres, and low water flow rates. The runner tip speed should be about one-half the hitting jet velocity for best efficiency. When running at 60-80 percent of full load, the efficiency (work generated by the turbine divided by the kinetic energy of the free jet) may surpass 91 percent. Using more than one jet may boost the power of a given wheel. For horizontal shafts, two-jet setups are popular. Two different runners are sometimes installed on one shaft, driving a single electric generator. Vertical-shaft engines may feature four or more distinct jets.

If the turbine's electric load varies, its power production must be quickly changed to meet the demand. To maintain the generator speed constant, the water flow rate must be changed. The flow rate via each nozzle is regulated by a hydraulic servomotor through a centrally positioned, finely formed spear or needle that glides forward or backward. Proper needle design ensures that the velocity of the water exiting the nozzle stays basically constant regardless of the opening, resulting in almost constant efficiency over much of the operational range. It is not advisable to cut water flow abruptly to match a drop in load. This may cause a damaging pressure spike (water hammer) in the supply pipeline or penstock. Such surges may be prevented by placing a temporary spill nozzle between the jet and the wheel, redirecting and dissipating some of the energy as the needle progressively closes from one side and continues in a single path, discharging on the other side. This turbine was employed in medium-sized units with somewhat high heads.

Turbines that react

Forces moving the rotor in a response turbine are generated by the reaction of an increasing water flow in the runner while the pressure declines. In a rotational lawn sprinkler, the emerging jet propels the rotor in the opposite direction, demonstrating the reaction principle. Reaction turbines may be employed across a considerably wider range of heads and flow rates than impulse turbines due to the wide range of available runner types. Reaction turbines generally include a spiral intake casing with control gates to manage the flow of water. As the flow increases, a portion of the potential energy of the water may be transformed to kinetic energy at the entrance. Following that, the water energy is extracted in the rotor. As previously stated, there are four basic types of reaction turbines in widespread use: the Kaplan, Francis, Deriaz, and propeller types. There is basically an axial flow through the machine in fixed-blade propeller and adjustable-blade Kaplan turbines (named after the Austrian inventor Victor Kaplan). The Francis- and Deriaz-type turbines (named after British-born American inventor James B. Francis and Swiss engineer Paul Deriaz, respectively) use "mixed flow," in which water enters radially inward and exits axially. The runner blades on Francis and propeller turbines are stationary, but the blades on Kaplan and Deriaz turbines may be rotated around their axis, which is at right angles to the main shaft.

Machines with axial flow

Fixed propeller turbines are often employed for big units with low heads, resulting in huge diameters and moderate rotational speeds. A propeller-type turbine runner, as the name implies, resembles a ship's extremely large propeller, except that it serves the opposite purpose: power is extracted in a turbine, while it is supplied into a maritime propeller. The propeller blades may be attached to the central shaft, or hub, during on-site construction, allowing transport by sections for a large runner. Vertical-shaft propeller turbines with low heads (less than 24 metres) often use a concrete spiral intake casing with a rectangular cross section. Inlet guiding vanes are either fixed on a ring or inserted separately into the concrete in big installations. Servomotor-driven wicket gates may enhance or reduce the flow passage. A draught tube, a conical diffusing departure section where the velocity is lowered while the pressure is raised, may partly recuperate the kinetic energy exiting the runner. This leads to increased efficiency by minimising kinetic energy loss at the installation's exit, or tail, portion.

In North America, where low heads and high flow rates are typical, propeller turbines are often employed. For example, the Moses-Saunders Power Dam on the St. Lawrence River between New York and Ontario has 32 propeller turbines, 16 of which are managed by the United States and 16 by Canada, with each turbine rated at 50,000 kilowatts. With such big plants, each turbine may be operated at or near its highest efficient output by switching full units in and out as the demand changes, in addition to controlling individual unit. If the head or flow rate varies annually, as it does in many river systems, an installation with just a few propeller turbines may be forced to run all units at partial power under typical flow and load circumstances. When the turbine load falls below 75% of its rating, the energy-conversion efficiency of a standard propeller turbine quickly diminishes. This performance loss may be reduced by adjusting the runner's inlet-blade angle to better match the runner-inlet conditions with the water velocity for a given flow. Each blade of a Kaplan turbine may be swivelled around a post at right angles to the main turbine shaft, resulting in variable pitch. An oil-pressure driven servomotor, normally placed in the rotor hub and supplied by the generator and turbine shaft, controls the angle of the blades. The servo-control system, which also drives the gates through a cam or rocker arrangement, is intended to alter angles and intake flows to fit the electrical demand while maintaining the main shaft and its directly connected generator running at a constant speed. Runners with four to six blades are usual, while higher heads may need more blades. Kaplan designs for heads up to 58 metres have been created by British manufacturers.

Although the typical turbine installation has a vertical shaft, some have horizontal shafts. The generator is buried in a nacelle, which corresponds to the thick body of a light bulb, and the blades are arranged around a hub, which corresponds to the thinner bulb socket, in a horizontal bulb configuration. This design is appropriate for medium-sized machines running at extremely low heads and requiring a nearly straight-through water flow.

Mixed-flow turbines

Francis turbines are perhaps the most often utilised because to their greater variety of appropriate heads, which typically vary from three to 600 metres. The flow rate and output must be high in the high-head range; otherwise, the runner becomes too narrow for acceptable manufacturing. Propeller turbines are often more efficient at low head, unless the power output is equally low.

Francis turbines, which are available in a variety of designs and sizes, rule supreme in the medium-head range of 120 to 300 metres. They may have horizontal or vertical shafts, with the latter being preferred for machines that diameters of two metres or more. Vertical-shaft machines often take up less space than horizontal systems, allow for more runner submergence with less deep excavation, and make the tip-mounted generator more readily accessible for servicing. Horizontal-shaft units are more compact for lower capacities and provide simpler access to the turbine, but removing the generator for maintenance becomes more difficult as size grows.

The most popular kind of Francis turbine features a spiral casing made of welded or cast steel. The casing uniformly distributes water to all input gates, with up to 24 pivoting gates or guiding vanes employed. Depending on the intended power output, the gates run from totally closed to wide open. Most are powered by a common regulating speed ring and therefore are pin-connected in such a way that if debris clogs one of the gate passageways, no harm occurs. One or two oil-pressure stepper motors controlled by the speed governor spin the regulating ring. Slow, high-power units feature virtually radial blades, while rapid, low-power units have curved blades that stretch from the radial input to nearly the axial outlet. Once the total blade specifications (inlet and exit diameters, as well as blade height) are determined, the blades are constructed to provide for a smooth entrance of the water flow at the inlet and a minimum of water swirl at the exit. The number of blades may range from seven to nineteen. Runners for low-head units are typically built of cast mild steel, with stainless-steel shielding added at cavitation-prone places (see below). For high heads, all stainless-steel construction is more typically employed. Large units may be welded together on-site by combining several prepared steel pieces to give precisely shaped, completed water passageways. Francis turbines enable the construction of extremely large, high-output machines. The Grand Coulee hydroelectric power facility on Washington's Columbia River contains the biggest single runner in the country, capable of generating 716,000 kilowatts at a head of 93 metres. The Itaip plant, located on the Paraná River between Brazil and Paraguay, is equipped with 18 Francis turbines capable of generating 740,000 kilowatts apiece with heads ranging from 118.4 to 126.7 metres while turning with little more than 90 revolutions per minute (rpm).

Deriaz mixed-flow turbines have swivelled, variable-pitch runner blades that increase efficiency at part loads in medium-sized equipment. The Deriaz design has proven effective for higher head applications as well as certain pumped storage applications (see below). It has a lower runaway (sudden loss of load) speed than a Kaplan turbine, which results in substantial cost savings for the generator. However, very few Deriaz turbines have been produced. At 1958, the first non-reversible Deriaz turbine was erected in an underground station in Culligran, Scotland, capable of producing 22,750 kwh with a head of 55 metres.

CHAPTER 9

BOUNDARY LAYER THICKNESS

Ramachandran T,
Professor, Department of Mechanical Engineering, Jain (Deemed-to-be University) Bangalore, India
Email Id-t.ramachandran@jainuniversity.ac.in

The thickness of the boundary layer is the angle measured from the solid surface at which the fluid's velocity equals 0.99 times the free stream fluid velocity approaching the solid surface.It is sometimes referred to as the perpendicular distance between boundary layer and the solid surface, where the boundary layer is the locus of all sites where the velocity is 0.99 times the free stream velocity (u or U).

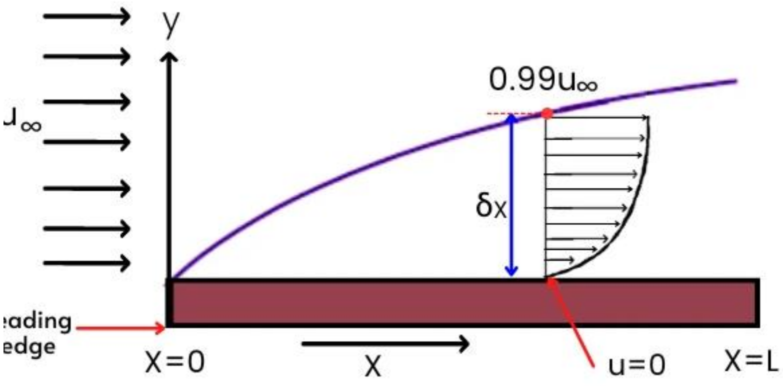

Figure 9.1 Boundary layer thickness.

It is represented by the symbol, and its value varies along the length of the solid surface. The graphic below depicts the formation of the shear layer over a flat solid surface. The boundary layer area is split into the following regions along the length of the plate. Laminar, turbulent, and transitional flow, in which laminar flow transitions to turbulent flow. Figure 9.1 Boundary layer thickness.

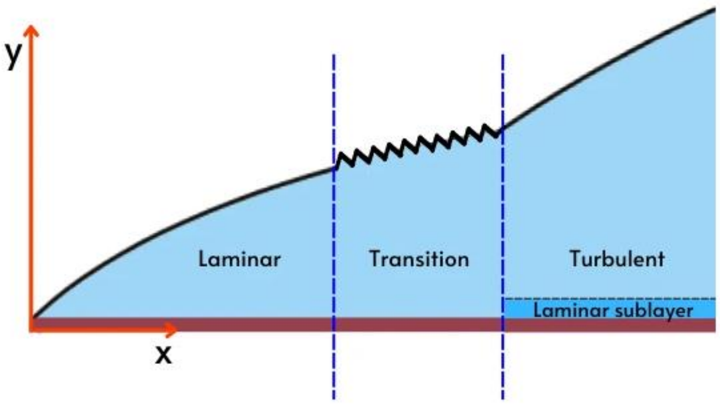

Figure 9.2 laminarr sublayer

The relevant boundary layer thickness in each location is termed as laminar boundary layer thickness and turbulent border layer thickness based on it. The turbulent zone also contains a smaller laminar area near the flat substrate known as the laminar sublayer, whose thickness is given by '. Figure 9.2 laminarr sublayer

Boundary layer thickness

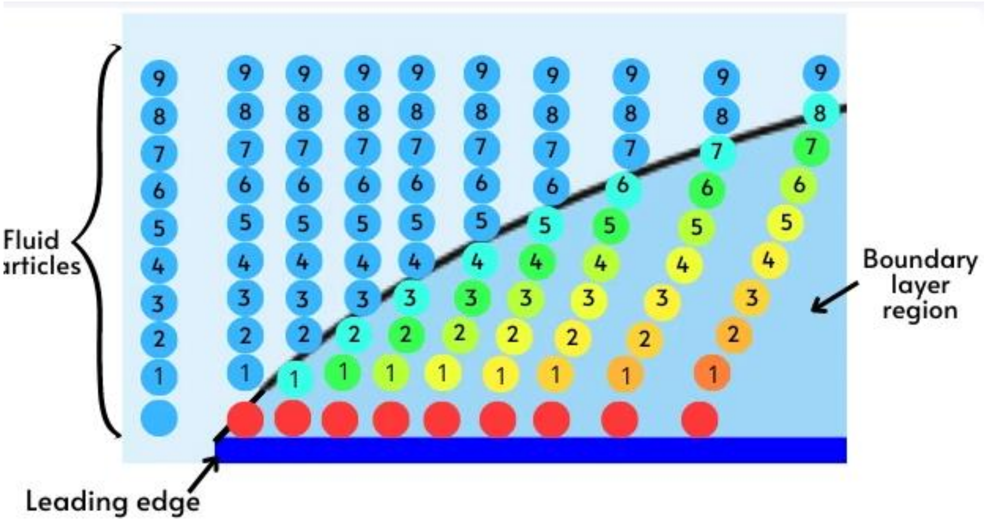

Figure 9.3 Boundary layer thickness increseased

Consider a fluid flowing in a laminar flow across a flat plate. The fluid particles are travelling with a constant free-stream velocity of 'U' before hitting the fixed solid plate. Figure 9.3 Boundary layer thickness increseased. When the liquids in the ottom layer come into contact with the leading edge of the plate, the fluid particles in the bottom layer cling to the plate owing to adhesion.

The particles in the bottom layer attract the fluid components in the adjacent higher layer due to the cohesive force between the fluid particles (viscosity). As a result, the molecules in the first layer are somewhat slowed. As the fluid passes over the plate, individual molecules in the bottom layer experience more retardation, causing the molecules of the first layer to experience greater retardation. The molecules in the first layer somewhat retard the molecules in the second layer due to the increase in retardation. As the molecules in the second layer progress farther, they experience additional retardation, which affects the ions in the layer above them. As the fluid progresses on the solid surface, the viscosity affects the mobility of more and more fluid layers. As a result, additional fluid layers penetrate the boundary layer area. As a result, the depth of the boundary layer rises continuously over the surface.

Pressure gradient

The pressure gradient depicts the rate of pressure change along the x-axis (dP/dx).If [dP/dx 0], the pressure is dropping along the flow direction, indicating that the kinetic energy is rising along the streamwise direction. It raises the pace along the flow direction, resulting in a reduction in the thickness of the boundary layer. If [dP/dx0], the pressure is rising along the flow direction. It implies that the fluid's kinetic energy is decreasing in the fluid flow. As a result, the fluid decelerates in the x-direction. We know that the fluid layers near the hard surface have the lowest velocity, and the unfavourable pressure impact may induce further slowdown of these bottom layers. As a consequence, the depth of the boundary layer increases.

Viscosity

The thickness of the boundary layer increases as the fluid's viscosity increases. Higher viscosity enables the fluid to retard more across the solid surface, resulting in increased boundary layer thickness. Thus, a fluid with a greater viscosity may form a thicker boundary layer than a fluid with a lower viscosity.

Fluid flow along an interior wall that induces a pressure impact on the fluid flow along the wall under examination is referred to as bounded boundary layers. The velocity profile corresponding to the wall generally smoothly different stages to a constant velocity value designated as ue, which is the distinguishing feature of this kind of boundary layer (x). Figure 1 depicts the bounded boundary layer idea for steady flow entering the bottom half of a thin flat plate 2-D channel with height H (the flow and plate extend in the positive/negative direction perpendicular to the x-y-plane). Fluid flow through most pipelines, channels, and wind tunnels exhibits this form of boundary layer flow. Figure 1 depicts a stationary 2-D channel with fluid flowing down the interior wall at a time-averaged velocity u(x,y), where x is the flow direction and y is the normal to the wall. The H/2 dashed line indicates that this is an internal pipe or channel flow condition with a top wall placed above the shown bottom wall. Figure 1 displays flow behaviour for H values greater than the maximum thickness of the boundary layer but less than the thickness at which the flow begins to behave as an external flow. If the wall-to-wall distance, H, is smaller than the viscous boundary layer thickness, the velocity profile, given as u(x,y) at x for all y, assumes a parabolic profile in the y-direction and the boundary layer thickness is simply H/2.

The fluid has zero velocity at the plate's solid walls (no-slip boundary condition), but as you travel away from the wall, the flow velocity grows without peaking and ultimately reaches a constant mean velocity ue (x). Depending on the shape of the wall, this asymptotic velocity may or may not vary along it. The boundary layer thickness is the point at which the velocity profile effectively achieves the asymptotic velocity. Figure 1 depicts the boundary layer thickness as a curving dashed line beginning at the channel entrance. It is hard to pinpoint the precise point at which the velocity profile achieves asymptotic velocity. As a consequence, a variety of boundary layer thickness parameters, indicated typically as ( ) delta (x), are employed to characterise distinctive thickness scales in the boundary layer area. The velocity profile shape is also of relevance for distinguishing laminar from boundary layer turbulent flows. The profile specific concerns to the velocity profile's y-behavior as it transitions to ue (x). Figure 9.4 Viscosity fluctuations of fluid

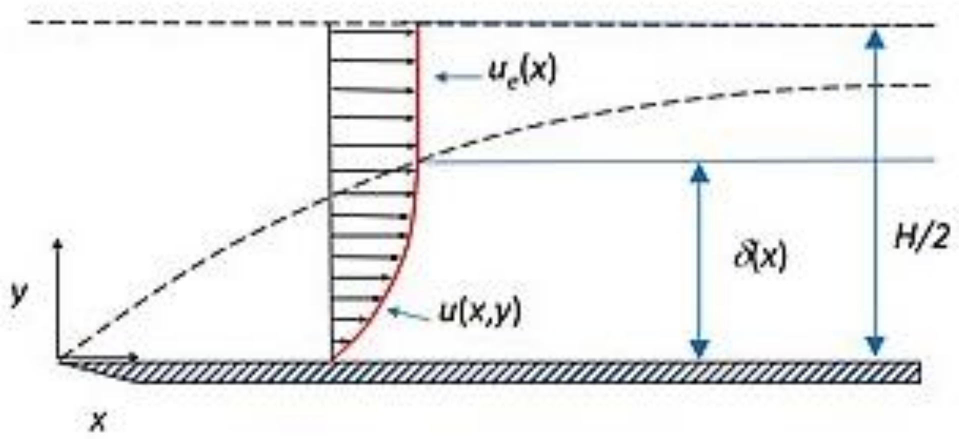

Figure 9.4 Viscosity fluctuations of fluid

Unbounded Boundary Layer

As the name indicates, unbounded boundary layers are often outer boundary layer flows along walls (and some very large gap interior flows in channels and pipes). The distinguishing feature of this kind of flow, which is not frequently recognised, is that the velocity profile passes through a peak around the viscous boundary layer border and then gently asymptotes here to free stream velocity u0. Near-wall air flow over a wing in flight is an example of this sort of boundary layer flow. Figure below depicts the unbounded boundary layer idea for constant laminar flow along a flat plate. The bottom dashed curve indicates the highest velocity umax(x), while the upper dashed curve shows the boundary layer thickness location, where u(x,y) effectively becomes u0. Because the peak is modest in the extremely thin flat plate case, the flat plate external boundary layer closely resembles the inner flow flat channel case. As a result, most of the liquid flow literature treats the constrained and unbounded instances as comparable.

The issue with this equivalency thinking is that the highest peak value for flow along a wing in flight may easily reach 10-15% of u0. In a series of Air Force Reports, the distinctions between the limited and unbounded boundary layers were investigated. Because of the unbounded boundary layer peak, several of the velocity profile thickness and shape parameters utilised in inner bounded boundary layer flows must be altered for this scenario. The laminar unbounded boundary layer situation, among other distinctions, features viscous and inertial dominated areas comparable to turbulent boundary layer flows (Figure 9.5)

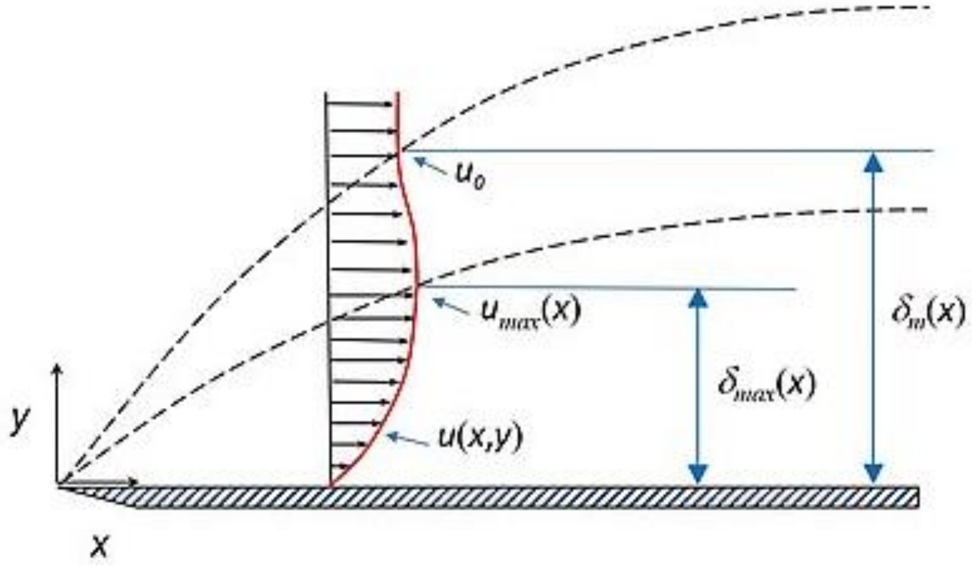

Figure 9.5 Unbounded Boundary Layer

Similarity and Similitude

The notion that the answer may be reached by performing experiments on comparable but not equal systems is one of the main points of dimensional analysis. The research presented here indicates and illustrates that the answer is based on a number of dimensionless numbers. As a result, building trials using the same factors may, in principle, give a solution to the issue at hand. Thus, understanding what dimensionless parameters are should offer information of how to build the experiments. This section discusses these similarities, which are referred to as analogy or similitude in the literature. It is difficult to achieve full resemblance. As a result, there is debate on how comparable the prototype is to the original. It is customary to distinguish three types of similarities: geometric, kinetic, and dynamic. This classification began for historical reasons, and it has some value, particularly when using Buckingham's technique. This distinction is less essential in Nusselt's technique.

Geometric Similarity

One logical aspect of dimensional analysis is that the experiences should be comparable to the real body that they are meant to reflect. This logical conclusion is an afterthought, and the author is unaware of any evidence for it based on Buckingham's methodologies. This result, ironically, is based on Nusselt's technique, which requires the same boundary conditions boundary constraints. Again, Nusselt's technique needs similarity because of the demands to the boundary conditions. This suggested concept is adopted here. According to this concept, the prototype area must be a square of the real model. They are the usual dimensions in two directions, subscript p refers to the prototype, and m refers to the model. Volumes fluctuate with length cubes under the same reasoning. In rare cases, the model cannot match two or so more dimensionless parameters. In such scenario, sacrificing geometric similarity to reduce the unwanted impacts is the answer. River modelling, for example, necessitates distorting vertical scales to avoid the impact of surface tension, bed roughness, or sedimentation.

Kinematic Similarity

When there is geometrical similarity and the movements of the fluid above the objects are the same, complete kinetics similarity is established. If this resemblance cannot be achieved, then the objective to generate a motion "image" defined by ratios of related velocities and vibration is consistent across the real flow field. In the literature, it is common to examine instances in which the model and prototype are comparable but the velocities vary due to a differing scaling factor. Aside from the forms and counters of the item, geometrical similarities may necessitate surface roughness and degradation of mobile surface surfaces or sedimentation of particles surface tensions. These establish requirements for a minimum friction velocity. In certain circumstances, the minimum velocity is Umin=w/. Thin film flow, for example, cannot obtain a low Reynolds number.

Dynamics Similarity

In the literature, there are numerous different and contradictory definitions of dynamic similarity. This word refers to the similarity of something like the forces in this context. Based on Newton's second rule, this necessitates similarity in accelerations and masses between model and prototype answer is a function of a number of common dimensionless factors. The Froude number is one such dimensionless parameter. Because both examples have the same Froude number, the solution for the model and the prototype is the same. As a result, it is possible to write that

$$\left(\frac{U^2}{g\ell}\right)_m = \left(\frac{U^2}{g\ell}\right)_p$$

It can be noticed that $t \sim \ell/U$ thus equation can be written as

$$\left(\frac{U}{gt}\right)_m = \left(\frac{U}{gt}\right)_p$$

Similarity Law Fluid Mechanics

Similarity rules are ideas that may be used to assess engineering models. A model is considered to be comparable to the real instance if the two share geometric, kinematic, and dynamic similarities. Similitude may be used in place of similarity in certain cases. The word dynamic similarity indicates the presence of geometric and kinematic similarities The principal use of similarities is in hydraulic and aeronautical engineering, where scaled models are used to evaluate fluid flow conditions. Furthermore, in many fluid mechanics textbooks, similarity is a key principle from which formulae are derived. Similarity rules, for example, are a good technique for categorising turbopumps in the same families. This categorization assists designers in designing comparable pumps using the same information. It also serves as a handy tool for turbomachinery users in selecting the optimum machine for specific application and customising it to particular circumstances.

Conceptual Basics of Similarity

Engineering models may be created to investigate challenging fluid dynamics issues when calculations and simulators are insufficient. Models are often, but not always, smaller than the real design. Scaled models enable the fabrication of a design and, in many circumstances, are critical phases in the development process. However, the building of a laboratory model must be accompanied by an analysis to establish the circumstances under which it will be evaluated. While the geometry of the entire model is readily scaled, other factors like as pressure, temperature, velocity, and fluid type may need to be altered. When the test settings are designed in such a manner that the test results are relevant to the actual model, similarity is obtained.

Geometric Similarity

The scaled model is shaped similarly to the application. More specifically, the model may be derived from the real instance using uniform scaling (enlarging or reducing). All circles, for example, are geometrically identical to one another. All squares and equilateral triangles are the same. On the other hand, ellipses, rectangles, and even isosceles triangles are not all the same. If the two angles of one triangle are equal to the two angles of another, the triangles are comparable.

Similarity in Kinematics

Kinematic similarity indicates that the velocity at each place in the model's flow is proportional to the velocity at the prototype's homologous location by a constant scale factor. As a result, it maintains the same flow streamline pattern. This is an essential requirement for the model and prototype to be completely comparable. In other words, kinematic similarity refers to the fluid's motion being comparable. Because movements may be represented in terms of distance and time, it implies length similarity (geometrical similarity) and time interval similarity. Dimensionless groups are studied in the science of fluid dynamics to achieve kinematic similarity in a scaled model. In many studies, for example, the Reynolds number of the model and the prototype must be equal. There are additional dimensionless numbers to consider, which we will go over in further depth in the sections that follow. Figure 9.6 Geometric Similarity.

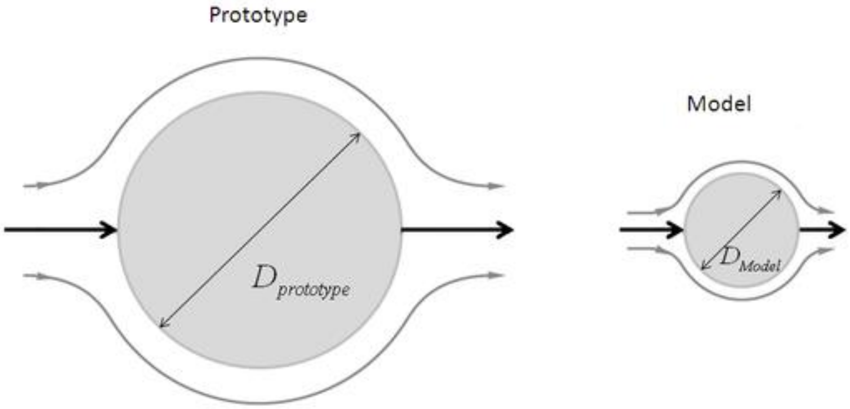

Figure 9.6 Geometric Similarity.

Dynamic Similarity

Dynamic similarity in fluid mechanics indicates that when two geometrically similar things have the same forms and different shapes, the same eqn, and equal dimensionless numbers, the fluid flows will be the same. The distribution of the velocity and pressure fields for any variation of flow may be derived by evaluating the fundamental Navier-Stokes equation with geometrically comparable bodies and equal dimensionless numbers. Using continuum mechanics concepts, all of the parameters required to characterise the system are identified.Dimensional analysis is used to investigate the system as thoroughly as feasible with the fewest independent factors and the greatest amount of dimensionless numbers.

The dimensionless parameters have the same values for the large scale study and the prototype. Because they are dimensionless, they may assure dynamic resemblance between the model and the real instance. The equations that arise are used to create scaling rules, which govern model testing conditions. It is frequently hard to create an exact likeness during a model test. The more unlike it is to the operational circumstances, the more difficult it is to establish resemblance. In these circumstances, certain characteristics of similarities may be neglected in favour of concentrating just on the most crucial factors.

It is very difficult to establish dynamic similarity for a partly submerged vessel while designing a maritime vessel; on the one hand, it is impacted by air forces above it. It is, however, impacted by the aerodynamic force of the air underneath it. Furthermore, the velocity of the waves between air and water surfaces influences its performance. Because the scale requirements for each of these events varies, models cannot perfectly recreate what occurs to a real vessel. However, since an aeroplane or submarine operates entirely inside one medium, it is possible to do so.

Dimensional Analysis

Dimensional analysis is the process of identifying the base quantities of various physical quantities (such as length, time, mass, and electric current) and units of measure (such as kilometres and kilogrammes) and tracking these dimensions to make calculations or comparisons in engineering and science.

Each equation or inequality must have the same dimensions on both the left and right sides. Dimensional homogeneity is the name given to this characteristic. A crucial step in dimensional analysis is to check for this simple rule. Many parameters in physics and engineering are numerically represented (a numerical value and a related dimensional unit). Normally, a quantity is expressed in terms of other quantities. The comparable unit for speed, for example, is a mix of length and time (e.g., measured in metres per second). Other types of relationships include multiplication (typically represented by a centred dot), powers (such as m2, square metres), and their combinations. A set of base units for a measuring system is a set of units selected such that none of them can be stated in terms of others, and all remaining units may be defined based on their combinations. Units of length and time, for example, are often used as basic units. As a result, the volume of a system may be measured using the basic length units (m3).

Buckingham π Theorem

The Buckingham theorem demonstrates that the validity of physical laws is not contingent on a certain unit system. This theorem implies that each physical law may be expressed as an identity containing only dimensionless combinations of the variables connected with the law; for example, volume & pressure are inversely linked by Boyle's Law. If the values of dimensionless combinations vary with the system of units, the equation is no longer an identity and the Buckingham theorem is no longer secure. This theorem aids in the discovery of a number of separate dimensionless quantities. Assume that a physical issue has a specific number (n) of variables, including m separate factors. As a result, the number of independent dimensionless quantities increases.

Dimensionless Numbers in Fluid Mechanics

Some important dimensionless numbers used in fluid mechanics and their importance is explained below.

1. Reynolds Number
2. Froude Number
3. Weber Number
4. Mach Number
5. Euler’s Number

Reynolds Number

The Reynolds number is the ratio of inertia to viscous force. It depicts the dominance of inertia forces over viscous forces in flow systems.

$$R_e = \frac{\rho . v . d}{\mu}$$

Where,

μ = viscosity of fluid (kg/m.s)

d = diameter of pipe (m)

v = velocity of flow (m/s)

Importance

The Reynolds number is relevant to both closed and free surface flows. Incompressible flow via tiny pipes, the motion of a submarine entirely submerged, flow through low-speed centrifugal compressor etc. are examples of applications where Reynolds number is important for determining flow behavior.

Froude number

The Froude number is defined as the ratio of force generated to gravitational force. The Froude number is important in free surface flows when gravity force dominates over other forces.

$$F_r = \frac{v}{\sqrt{g.L}}$$

Where,

L = flow length (m)

v = flow velocity (m/s)

g = gravity acceleration (m/s2)

The Froude number may be used to explain flow in open channels, flow through notches and weirs, ship motion in stormy wave action (ship resistance), flow over spillways, and so on.

Weber number

The ratio of inertia force to surface tension is known as the Weber number. Surface tension is generally involved in the creation of droplets or freshwater bubbles in a fluid. Surface tension increases when the Weber number decreases, and vice versa.

$$W_e = \frac{\rho . d . v^2}{\sigma}$$

When surface tension is dominating, the Weber number is less than one. It occurs when the liquid surface's curvature is minimal in comparison to its depth. This may be observed in a variety of scenarios, including blood flow in veins and arteries, liquid atomization, capillary water circulation in soils, thin sheets of fluid moving over a surface, and so on.

Mach number

The Mach number is the ratio of inertia to elastic force. If the Nusselt number is one, the flow velocity equals the sound velocity in the fluid. If it is less than one, the flow is known as subsonic flow; if it is larger than one, the flow is known as supersonic flow.

$$M_a = \frac{v}{c}$$

Where,

v = Flow velocity (m/s) and c = Sound velocity in a fluid (m/s)

The Mach number owing to the local speed of sound is affected by the surrounding media at different temperatures and pressures. The Mach number may be used to assess whether a flow is incompressible or compressible. The medium might be either liquid or gas. The medium may be moving while the boundary is steady, or the boundary may be travelling in a resting media. Both the medium and the border may be moving at a particular speed, but their relative velocities matter. The medium may be channelled via various devices such as wind tunnel testing or submerged in it. Because it is a mixture of two speeds, the Mach number is known as a dimensionless number.

Euler's Number

Euler's number (e) is a mathematical expression that represents the base of the natural logarithm. This is represented as a never-ending non-repeating number. Euler's number has the first few digits 2.71828. The number is widely employed in situations involving exponential growth or decay and is generally symbolised by the letter e. Euler's number may alternatively be seen as the foundation for an exponential function whose value is always comparable to its derivative. In other words, e is the only integer such that ex grows at a rate of ex for any conceivable x. The base of the natural logarithm is expressed using uler's number. E is a set of numbers that starts with 2.71828. It, like pi, is non-terminating, which means it continues indefinitely. It's also an irrational number, thus it can't be stated as a fraction. It may be used to compute the decay or increase of a certain component over time, such as compounding. Consider lending money at a 100% annual compounded interest rate. Your money would have doubled in a year. The overall returns increase modestly as the interval narrows. If interest is computed n times each year at a rate of 100%/n, the total accumulated wealth at the conclusion of the first year is slightly more than 2.7 times the original investment if n is big enough.

History of Euler's Number

Although it is most often identified with and named after Swiss mathematician Leonhard Euler, it was discovered in 1683 by physicist Jacob Bernoulli. He was attempting to calculate how wealth would expand if interest was compounded more often rather than annually. Leonhard Euler did not complete the most important study on the number until many decades later. Euler demonstrated in his work Introductio in Analysin Infinitorum (1748) that it was an irrational number whose digits would never repeat. He also demonstrated that the number is an endless sum of reverse factorials:

Euler employed the letter e for multipliers, but the letter has become synonymous with his name. It is widely employed in a variety of applications, including population expansion in living creatures and nuclear scientists' radioactive decay of heavy substances such as uranium. It is also useful in geometry, probability, and other branches of applied mathematics. Euler's number is widely used in issues involving growth or disintegration, where the rate of change is dictated by the present value of the number under consideration. In biology, for example, bacterial populations are predicted to double at regular periods. Another example is radiometric dating, in which the number of radionuclides is projected to decrease during the element's specified half-life.

Navier-stokes equation

In certain cases, 11 grows as x1 increases. The force exerted by this component of stress on the right-hand side of the cubic element of fluid shown in Figure 9B will therefore be higher than the force exerted on the opposite side, and the difference between the two will drive the fluids to accelerate along x1. If 12 and 13 grow with x2 and x3, respectively, accelerations along x1 will occur. The fluid's equation of motion describes these accelerations, as well as the analogous accelerations in the other two directions. This equation has the form Equations for a fluid flowing so slowly relative to the rate of sound that it may be viewed as incompressible and where temperature fluctuations from place to place are inadequate to induce major differences in shear viscosity. Euler determined all of the terms in this equation excepting the one on the left-hand side proportionate, and the equation is known as the Euler equation without that term. The complete thing is known as the Navier-Stokes equation.

The equation is stated in a compact vector notation that many readers will find completely incomprehensible; nevertheless, a few words of explanation may aid some others. The symbol denotes the gradient function, which, when followed by a scalar number X, produces a vector of components (X/x1, X/x2, X/x3). The vector product of this operator and the fluid velocity v, denoted by (v), is commonly referred to as curl v [and (v) is also curl curl v]. Vorticity is another term for (v), which conveys the features of the local flow pattern that it depicts especially clearly. The vorticity in a sample of fluid spinning like a solid body with uniform angular velocity 0 is in the same direction as the axis of rotation, and its magnitude is equal to 20. In other cases, the vorticity is connected to the local angular velocity in a similar way and might vary from place to place.

On the right side of (155), Dv/Dt indicates the rate of change of velocity that would be seen if the motion of a single element of the fluid could be abided is, the element's acceleration while v/t reflects the rate of change at a fixed location in space. If the flow is constant, v/t is always zero, but the fluid may be accelerating when individual fluid elements migrate from areas where the streamlines are widely spread to regions where they are close combined. The disparity between Dv/Dt and v/t—i.e., the last (v)v component in (155) is what adds the nonlinearity that makes fluid dynamics so unpredictable.

Stokes’s law

Stokes' law is a mathematical equation that represents the drag force that prevents tiny spherical particles from falling through a fluid medium. The rule, initially proposed in 1851 by British physicist Sir George G. Stokes, is obtained by considering the forces acting on a specific particle as it descends down a liquid column there under effect of gravity. The drag force F acting upward in opposition to the fall, according to Stokes' equation, is equal to 6rv, where r is the radius of the sphere, is the stickiness of the liquid, and v is the pace of fall. The downward force is equal to 4/3r3 (d1 d2)g, where d1 is the density of the sphere, d2 is the concentration of the liquid, and g is the gravity acceleration. The upward and downward forces are balanced at a constant velocity of descent known as the terminal velocity. Equating the two formulas provided above and solving for v gives the needed velocity as v = 2/9(d1 d2)gr2/. Stokes' law is used in a variety of situations, including the settling of silt in fresh water and the measuring of fluid viscosity. However, since its applicability is restricted to situations in which the particle's motion does not induce disturbance in the fluid, different changes have been proposed.

Potential flow with circulation: vortex lines

Thomson's theorem is based on the notion of circulation, which Thomson introduced. This number is specified for a closed loop immersed in and moving with the fluid; indicated by K, it is the integral of v dl around the loop, where dl is a length element along the loop. If the vorticity is 0 everywhere, then the circulation around all conceivable loops is also zero, and vice versa. Thomson demonstrated that K cannot change if the viscous factor in adds nothing to the local acceleration, and so K and vorticity stay constant throughout time.

The constant flow pattern described earlier may be created by spinning a cylindrical spindle in a fluid; the streamlines are circles from around spindle, and the velocity falls down like r1. This flow pattern occurs naturally in whirlpools and typhoons, where a "core" in which the fluid spins like a solid body serves as the spindle; the axis around which the fluid circulates is thus referred to as a vortex line. When evaluated in isolation for a brief period of time, each little piece of fluid outside the core seems to be experiencing translation without rotation, and the local vorticity is zero. If this were not the case, the viscous torques would not cancel and the flow pattern would not be stable. However, if the loop for which it is defined encloses the spindle or core, the circulation is not zero. In such cases, a potential that obeys Laplace's equation outside the spindle or core may be obtained, but it is no longer single-valued, to use a technical phrase that some readers may be acquainted with.

This phenomenon (called after the German physicist and chemist H.G. Magnus, who discovered it) occurs when fluid flows constantly through a cylindrical spindle, with a velocity that is perpendicular to the spindle's axis and evenly equal to, say, v0, while the spindle itself is slowly rotating. The fluid receives rotation, and the circulation around any loop that encloses the spindle (and encloses a layer of fluid next to the spindle within which the vorticity is nonzero and potential theory is inapplicable) has a nonzero value K in the steady state. depicts the streamlines that represent the constant flow pattern (apart from the "border layer"), however the details depend on the magnitudes of v0 and K. Because the pressure is strong at the stagnation locations in the flow pattern, the spindle should feel a downward force transverse to both its axis and the direction of v0.

This Magnus force is precisely comparable to the force exerted by a transverse magnetic field B0 on a wire carrying any electric current I, the amplitude of which equals B0I per unit length of the wire. The Magnus force on revolving cylinders has been used to propel experimental boats, and it is connected to the lift force on airfoils, which allows aircraft to fly (see below Lift). However, the transverse forces who cause spinning balls to veer in flight are not Magnus forces, as some claim. They are caused by the asymmetrical structure of eddies that form at the back of a whirling sphere (see below Boundary layers and separation). Cricket balls, unlike baseball, tennis, and golf balls, feature a high equatorial seam that contributes to the asymmetry of the eddies. In cricket, a bowler who wants to make the ball swerve puts spin on it, although he does it primarily to guarantee that the direction of the seam stays constant as the ball advances toward the batter.

Straight vortex lines of equal but opposing intensity, K, parallel and separated by a distance d, would drift sideways across the fluid at a speed indicated by K/2d, as proven by the magnetic counterpart or in other methods. Similarly, a vortex line that has linked up on itself to create a closed vortex ring at radius a drifts around its axis with a speed defined by where c is the radius of the line's core and ln stands for natural logarithm. This formula, for example, applies to smoke rings. The slowing of such rings as they propagate may be explained by the increasing of c with time due to viscosity.

CHAPTER 10

DRAG AND LIFT

Ramachandran T,
Professor, Department of Mechanical Engineering, Jain (Deemed-to-be University) Bangalore, India
Email Id-t.ramachandran@jainuniversity.ac.in

A fluid stream produces a drag force FD on every obstruction in its path, and the same force is produced if the obstacle moves while the fluid remains stationary. How huge it is and how it may be minimised are clear concerns for designers of moving vehicles of all kinds, as well as designers of refrigeration systems and other buildings who want to ensure that the structures do not collapse in the face of strong winds. Stokes originally produced an estimate for the drag force on a sphere that is valid at such low velocities that the v2 term with in Navier-Stokes equation is insignificant, and therefore at velocities where the boundary layer thickness indicated by (171) is greater than the sphere diameter D. It is also known as Stokes' law and may be written as Equation. One-third of this force is communicated to the sphere via shear stresses around the equator, with the remaining two-thirds owing to greater pressure at the front of the sphere than at the back.

As the velocity rises and the thickness of the boundary layer diminishes, the influence of shear stresses (or what is often referred to as skin friction in this context) becomes more and more essential in comparison to the effect of the pressure differential. It is hard to quantify that difference exactly, especially in the limit to which Stokes' rule applies, although once eddies emerge, it is likely to be about v02/2. At high speeds, one may predict Equation, where A′ is some effective cross-sectional area, probably equivalent to its real cross-sectional area A (D2/4 for a sphere), but not necessarily precisely equal to this. It is common practise to express drag forces in terms of such a dimensionless variable known as the drag coefficient, which is defined as the ratio [FD/(v02/2)A] and indicated by CD regardless of the form of the body. At high speeds, CD is plainly the same as the ratio (A′/A) and should hence be of order unity.

This is the limit of theory for this issue. Dimensional analysis principles can be used to demonstrate that, if the fluid's compressibility is irrelevant (i.e., if the flow velocity is far below the speed of sound), the drag coefficient must be some universal role of another nondimensional standard compound as the Reynolds number and defined as Equation. However, in order to identify the shape of this function, one needs conduct trials. Because the function is universal, a small number of trials will sufficient. They may be carried out using whichever liquids and spheres are most convenient, as long as the whole range of R that is expected to be significant is covered. Once the data have been displayed on a CD versus R graph, the graph may be used to forecast the drag forces experienced by other spheres in other liquids at velocities that may vary significantly from those used so far. This fact is worth highlighting since it enshrines the idea of dynamic likeness, which engineers significantly rely on when using model findings to anticipate the behaviour of much bigger systems.

Stokes' law is represented by this same straight line on the left of when stated in terms of CD and R. When R surpasses roughly 1, this law clearly fails. There is a significant range of R in the centre of the picture where CD is around 0.5, but as R reaches approximately 3 105, it drops rapidly to about 0.1. curves for cylinders of diameter D with transverse axes to the flow direction and transverse discs of dimension D. The cylinder curve is comparable to the sphere curve (albeit there is no straight-line section at low Reynolds numbers to match to Stokes' law), but the disc curve is much flatter. This flatness is due to the fact that a disc has sharp edges around which streamlines swiftly converge and diverge. The enormous pressure gradients that develop at the edge encourage the production and shedding of eddies. By assuming a drag coefficient of unity, the drag force on a transverse flat plate of any form can typically be predicted relatively precisely, provided its edges are sharp.

Because sharp edges encourage the creation and spilling of eddies, which increases the drag coefficient, streamlining the obstruction may minimise the drag coefficient. Separation happens at the back of the obstruction, hence it is the back that requires streamlining. The pressure gradient operating on the boundary layer behind the obstruction may be greatly lowered by extending this out as show several more techniques of lowering drag that have some practical uses. The obstruction in 17B is a wing with a slit in its leading edge; the current of air funnelled through this slot gives forward velocity to the fluid in the boundary layer on the top surface of the wing, preventing it from moving backward. Cowls, which are often affixed to the leading edges of aircraft wings, serve a similar role. the obstacle has an internal device—some sort of pump—that prevents the accumulation of boundary-layer fluid that would otherwise lead to separation by sucking it in through small holes in the surface of the obstacle near Q; the fluid can then be ejected through holes near P′, where it will do no harm.

It should be emphasised that the curves in Figure 16 are only universal if the velocity v0 is substantially smaller than the speed of sound. When v0 approaches the speed of sound, VS, the fluid's compressibility becomes important, which implies that the drag coefficient must be considered as depending on the dimensionless ratio M = v0/VS, also known as the Mach number, as well as the Reynolds number. The drag coefficient always increases as M approaches unity, although it may then decrease. To minimise drag in the supersonic range, it is better to streamline objects or projectiles from the front rather than the back, since this lowers the intensity of the shock cone.

Lift

If an aircraft wing, or airfoil, is to operate properly, it must encounter both an upward lift force and a drag force while the aircraft is in motion. The lift force exists because the displaced air flows faster over the top of the airfoil (and over the top of the associated boundary layer) than it does over the bottom, and therefore the pressure acting on the airfoil from below is larger than the pressure acting from above. It may also be seen as an unavoidable result of the limited circulation that occurs around the airfoil. As mentioned before in the Magnus effect explanation, one approach to create circulation around an impediment is to spin it. The circulation around an airfoil, on the other hand, is caused by its forward motion; it begins when the airfoil travels quickly enough to shed its first eddy.

The lift force on an airfoil travelling through stationary air at a constant speed v0 is the same as the lift force on an identical airfoil moving through stationary air at a constant speed v0 in the other direction; the latter is simpler to show graphically. Figure 18A depicts a collection of streamlines reflecting prospective flow across a stationary inclined plate prior to the formation of any eddy. The pattern is symmetrical, and the pressure fluctuations that accompany it produce both drag nor lift. However, near the back of the plate, the streamlines diverge fast, creating conditions for the creation of an eddy with a anticlockwise spin. Because the plate's edges are sharp, it grows faster swiftly and sheds more quickly. Figure 18B depicts several streamlines for the same plate after shedding while the detached eddy, known as the beginning vortex, is still visible. The circulation around the closed loop depicted by a broken curve in this picture was zero before the eddy occurred, and it must still be zero according to Thomson's theorem (see above Potential flow). To compensate for circulation +K of the initial vortex, a vortex line passing around this loop must have clockwise circulation -K. This additional line, known as the bonded vortex, is not visible in the figure because it is connected to the plate and stays so when the beginning vortex is swept downstream. It does, however, appear as a change in the flow pattern just behind the plate, where the streamlines no longer diverge as seen in Figure 18A. Because the divergence has been removed, no new eddies are expected to arise.

The formula v0K was previously mentioned for the strength of the Magnus force per unit length of a spinning cylinder, and the same formula may be used to the inclined plate in Figure 18B or to any airfoil that has shed a beginning vortex and hence has circulation around it. The validity of the formula is unaffected by the particular shape of the airfoil, much as the force produced by a magnetic field on a wire carrying a current is unaffected by the cross-sectional shape of the wire. Nonetheless, the design of the airfoil has a significant impact on the size of the lift force since it dictates the magnitude of K. Figure 17B depicts the kind of cross section that is used for aircraft wings. For reasons that have previously been discussed, the rear edge is made as sharp as possible, and it may take the shape of movable flaps that are dropped upon takeoff. Lowering the flaps improves K and hence lift, but they must be raised after the aircraft has achieved cruising altitude since they produce unwanted drag. Improve the angle (see Figure 17B) at which the primary section of the airfoil is oriented to the direction of motion to increase circulation and lift. However, there is a limit to the amount of lift that can be created in this manner because if the inclination is too extreme, the boundary layer splits behind the leading edge of the wing and the confined vortex, on which the lift relies, may be shed as a consequence. The aeroplane is reported to stall at this point. To prevent stalling, the leading edge is designed as smooth and rounded as feasible.

Thomson's theorem may be used to demonstrate that if the airfoil is finite in length, the beginning vortex and the bound vortex must both be part of a single, continuous vortex ring. They are connected by two trailing vortices that travel rearward from the airfoil's ends. As time passes, these trailing vortices lengthen, requiring more and more energy to fuel the swirling motion of the fluid surrounding them. In the scenario when the airfoil is moving and the air is fixed, it is obvious that this energy can only originate from whatever agent forces the airfoil ahead, and so the following vortices are a source of extra drag. The size of the increased drag is proportional to K2, but it does not increase when the airfoil is made longer while K remains constant. As a result, designers who want to optimise the lift-to-drag ratio will build their aircraft's wings as long as they could to long as strength and stiffness standards allow. When a boat sails into the wind, its sail works as an airfoil, with the mast serving as the leading edge, and the same principles that favour wide wings for aeroplanes also favour tall masts.

Convection

convection is therefore produced, which increases the rate at which heat is lost from the radiator since it brings cooler air into contact with it. Once convection is created, heat loss is complicatedly dependent on the spacing between the plates (D) as well as the thermal diffusivity (), specific heat, density, thermal expansion coefficient (), and viscosity of the fluid. Of course, the heat loss relies on (T1 - T2), and it is worth noting that the relationship is not linear; the heat loss grows faster than the temperature difference. Newton's law of cooling, which postulates a linear connection, is followed only when convection is inhibited or induced (when a radiator is fan-assisted, for example).

Consider the identical two plates in a horizontal rather than vertical configuration. If the hot plate is above the cold one, no convection can occur, and it is not clear that it happens in the opposite condition. The size of the temperature differential is determined by a dimensionless combination of several of the important factors, gD3(T1 - T2)/, which is known as the Rayleigh number. If the Rayleigh number is less than 1,708, the fluid is stable (or, maybe more accurately, metastable), even if it is warmer at the bottom than at the top. When the number 1,708 is surpassed, a pattern of convective rolls known as Bénard cells forms between the plates. The regular columns of cloud that develop over rising air areas provide evidence for the presence of such cells in the convecting atmosphere. Their periodicity may be astoundingly consistent. Macroscopic convective instabilities, such as the creation of Bénard cells, are a hallmark of both the seas and the atmosphere, and are typically connected with salt gradients rather than temperature gradients. A rigorous examination of the Earth's atmospheric and oceanic circulation.

Impulse-Momentum Theorem

According to the impulse-momentum theorem, the impulse imparted to an item equals the change in its momentum. It demonstrates that the changes in momentum of an object is determined not just by the amount of force applied, but also by the length of time it is applied. Only the size of the power and the time interval may be altered in most circumstances when the change in momentum stays constant. Over example, a significant force applied for a short period of time might alter an object's change in momentum. The same momentum shift may be accomplished by increasing the duration of time at a lower force.

In athletics, the phrase momentum is often used. When a pundit says that a player has momentum, it suggests that the athlete is on the move and will be tough to stop. Because a body with momentum cannot be halted, it is necessary to exert a force against its direction of movement for a certain amount of time. The greater the momentum, the more difficult it is to halt. As a result, more effort is necessary and more time is required to bring it body to a stop. As the force works on the body for a specific period of time, the velocity of the body varies, and therefore the momentum of the body changes. A force may modify the velocity of an item in either direction. Furthermore, when the object's velocity varies, so does its momentum. The idea of momentum is a physics concept. Any item having momentum will be difficult to stop. To bring such an item to a halt, a force must be applied against its motion for a certain amount of time. The more momentum an item has, the more difficult it is to stop. As a result, bringing such an item to a standstill would need more power, more time, or both. As the force works on the item for a specific period of time, the velocity of the object changes, and therefore the momentum of the object changes.

The ideas in the preceding paragraph should not seem abstract to you. If you've ever watched football, you've probably seen this a few times. In football, defensive players use force for a certain period of time to halt the offensive player with the ball's momentum. You've probably seen this a hundred times while travelling. When you come to a stop in front of a stop sign or a stoplight, the brakes apply a force to the automobile for a certain period of time in order to modify the car's motion. If a force is supplied to a moving item for a certain period of time, it may be stopped.

A force applied for a certain length of time changes the momentum of an item. In other words, an uneven force always accelerates an item, either faster or slower. When a force operates in the opposite direction of an object's motion, it slows it down. When a force operates in the same direction as an object's motion, the object accelerates. A force will modify the velocity of an item in either case. And if the object's velocity changes, the object's momentum changes as well.

These ideas are just an extension of Newton's second law, which was covered in a previous section. Newton's second law (Fnet = m * a) states that an object's acceleration is directly proportional to the net force exerted on the object and inversely proportional to the object's mass. When the concept of acceleration (a = change in velocity / time) is used, the following equalities are obtained.

$F = m * a$

or

$F = m * \Delta v / t$

When both sides of the above equations are multiplied by the number t, a new equation is formed.

$F * t = m * \Delta v$

This equation reflects one of two key concepts that will be used in the collision analysis throughout this unit. To completely comprehend the equation, it is necessary to comprehend its meaning in words. In other words, the force multiplied by the duration equals the mass multiplied by the velocity change. The quantity Force * time is known as impulse in physics. And, because the amount m*v represents momentum, the quantity m*v must represent momentum change. The equation really states that the

Momentum change = Impulse

Understanding the mechanics of collisions is one of the unit's main goals. The physics of collisions is regulated by momentum rules, and the first law discussed in this section is given in the preceding equation. The impulse-momentum change equation is the name given to this equation. The law may be stated as follows:

An item in a collision receives a force for a specified period of time, resulting in a change in momentum. The force applied for the given period of time causes the object's mass to either accelerate or decelerate (or changes direction). The impulse felt by the object equals the item's change in momentum. F * t = m * v is the equation.

Objects in a collision receive an impulse; the impulse causes and equals the change in momentum. Consider a football halfback who is sprinting down the field and collides with a defensive back. The impact would alter the halfback's pace and, as a result, his momentum. If the motion were depicted by a ticker tape diagram, it may look like this:

The collision happens at around the tenth dot on the diagram and lasts for a given period of time; in terms of dots, the contact lasts for approximately nine dots. The halfback encounters a force that lasts a particular length of time in the halfback-defensive back collision to modify his momentum. Because the impact slows the right-moving halfback, the force on the halfback must have been directed leftward. If the halfback was subjected to a force of 800 N for 0.9 seconds, the impulse would be 720 N*s. This impulse would result in a 720 kg*m/s change in momentum. In a collision, an object's impulse is always equal to the momentum change.

Hydraulic networks are intended to transport water from a single source to several end users. These end customers will be situated in various areas within a certain region. What propels water from its source to the ultimate user? As you may be aware, in water distribution systems, we do not employ active equipment like as pumps to transport water from the source (ESR) to customers; instead, we use the energy stored/available in the water itself. Understanding the concepts of head and head loss is critical to comprehending how this is accomplished. Let us first comprehend the basic concept as stated by Daniel Bernoulli (1700 - 1782) in his 1738 work Hydrodynamica.

Orifice plate

The orifice plate is the most prevalent kind of pressure-based flow element. This is basically a metal plate with the a hole in the centre through which fluid may flow. Orifice plates are commonly placed between two pipe junction flanges, allowing for simple installation and removal (Figure 10.1).

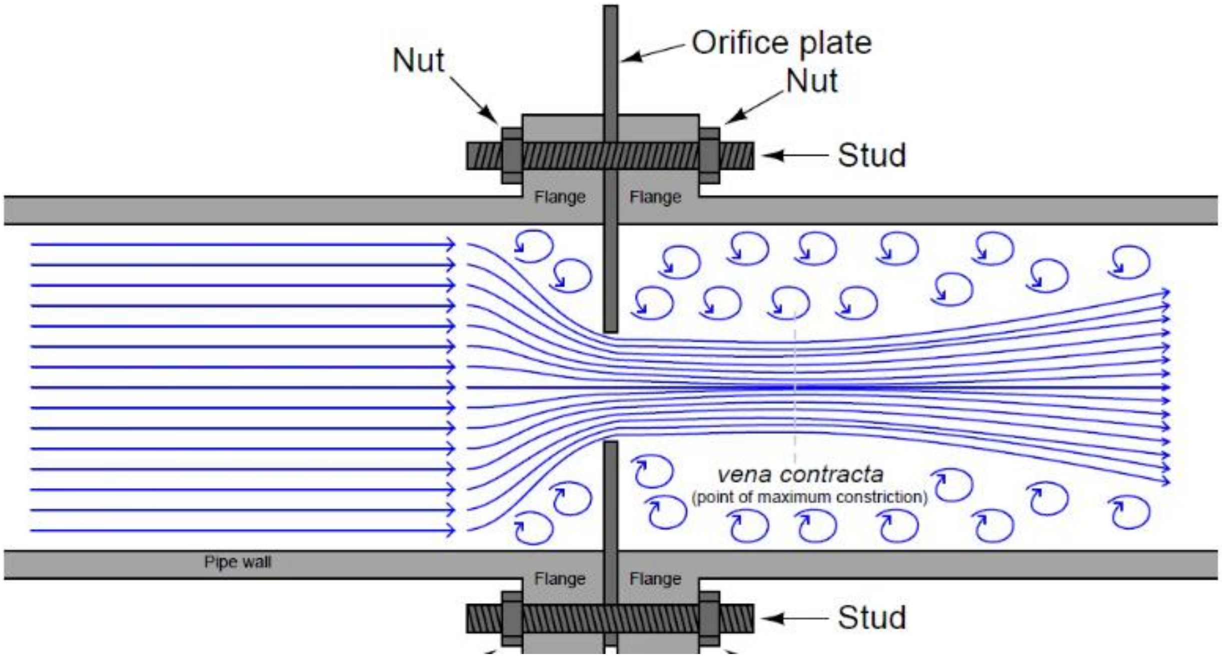

Figure 10.1 Impulse-Momentum Theorem

When the fluid is single-phase (rather than a mixture of gases and liquids or liquids and solids) and well-mixed, the flow is ongoing rather than pulsating, the fluid occupies the entire pipe (excluding silt or trapped gas), the flow profile is even and well-developed, and the fluid and flow rate meet certain other conditions, orifice plates are most commonly used to measure flow rates in pipes. Under these conditions, and when the orifice plate is built and fitted in accordance with industry, national, and international standards, the flow rate may be simply estimated using known equations based on extensive research that published in industry, national, and worldwide standards.

A calibrated orifice plate is one that has been calibrated using an appropriate fluid flow and a traceable flow measuring instrument.

Plates are often manufactured with sharp-edged circular orifices and mounted concentric with the pipe, with pressure tappings at one of three standard pairs of distances upstream and downstream of the plate; ISO 5167 and other important standards cover these kinds. There are plenty alternative options. The edges may be rounded or conical, the plate can have an aperture the same size as the pipe except for a segment at the top or bottom that is blocked, the orifice can be eccentric to the pipe, and the pressure tappings can be in different places. Various standards and handbooks cover variations on these possibilities. Each combination produces varied discharge coefficients that may be anticipated as long as certain requirements are satisfied, conditions that range from one kind to the next. Once the orifice plate has been constructed and fitted, the flow rate can typically be determined with a reasonable degree of precision by calculating the square root of the differential pressure across the orifice's pressure tappings and using an appropriate constant. Orifice plates are also used to lower pressure or limit flow, and are also referred to as restriction plates. Pressure tappings (also known as taps) have three conventional locations, which are generally referred to as:

Corner taps located immediately upstream and downstream of the plate; useful when the plate includes an orifice carrier with tappings D and D/2 taps or radius taps located one pipe diameter surface water and half a pipe diameter downstream of the plate; these may be installed by welding bosses to the pipe.

Flange taps are typically located 25.4 mm (1 inch) upstream and downstream of the plate, inside specialised pipe flanges. ISO 5167 and other significant standards cover these categories. 12D and 8D taps or recovery taps installed 2.5 pipe dimensions upstream and 8 diameters downstream, where the measured difference equals the unrecoverable pressure loss produced by the orifice.

Vena contracta tappings are located in the plane of minimum fluid pressure one pipe diameter upstream and 0.3 to 0.9 diameters downstream, depending on the orifice type and size relative to the pipe. Because the recorded differential pressure vary for each combination, the coefficient of discharge utilised in flow calculations is partially determined by the tapping sites. The most basic installations employ single tappings upstream and downstream, but these may be unreliable in certain cases; they can be blocked by particles or gas bubbles, or the flow profile might be uneven, causing pressures at the tappings to be higher or lower than the average in those planes. Multiple tappings circumferentially around the pipe and linked by a piezometer ring, or (in the case of corner taps) annular slots extending entirely around the internal circumference of the orifice carrier, may be employed in these instances.

Plate standards and handbooks are mostly concerned with thin plates with sharp edges. The leading edge of these is sharp and burr-free, and the cylindrical section of the orifice is short, either because the whole plate is thin or because the downstream edge of the plate is bevelled. The quarter-circle or quadrant-edge orifice, which has a completely rounded leading edge but no cylindrical part, and the conical inlet or conical entry plate, which has a bevelled leading edge but a very short cylindrical section, are exceptions. The orifices are typically concentric with the pipe (with the eccentric orifice being an exception) and round (except in the specific case of the segmental or chord orifice, in which the plate obstructs just a segment of the pipe). Standards and handbooks require that the plate's upstream surface be exceptionally flat and smooth. To enable condensate or gas bubbles to travel down the pipe, a tiny drain or vent hole is sometimes bored through the plate where it joins the pipe.

Pipe standards and handbooks provide for a well-developed flow profile, with velocities lower near the pipe wall than in the centre but no eccentric or jetting. Similarly, the flow downstream of the plate must be free of obstructions, otherwise the downstream pressure would be influenced. To do this, the pipe must be round, smooth, and straight across specified lengths. When providing enough straight pipe is not feasible, flow conditioners such as tube bundles or plates with many holes are introduced into the pipe to straighten and develop the flow profile, although even these need an additional length of straight pipe before the orifice itself. Some standards and handbooks also allow for flows from or into broad expanses rather than pipes, with the condition that the region before or after the plate be free of obstructions and flow anomalies.

Hagen–Poiseuille equation

The Hagen-Poiseuille equation, commonly known as the Poiseuille law or Poiseuille equation in nonideal fluid dynamics, is a physical law that determines the pressure drop in an incompressible and Newtonian fluid in laminar flow flowing through a long cylindrical pipe with constant cross section. It has been used effectively to air flow in lung alveoli, as well as flow via a drinking straw or a hypodermic needle. It was independently discovered in 1838[1] by Jean Léonard Marie Poiseuille and Gotthilf Heinrich Ludwig Hagen, and published by Poiseuille in 1840-41 and 1846. George Stokes provided the theoretical explanation for the Poiseuille law in 1845.

The equation assumes that the fluid is incompressible and Newtonian, that the flow is laminar through a pipe with a constant circular cross-section that is significantly longer than its diameter, and that there is no fluid acceleration in the pipe. Actual fluid flow is turbulent at velocities and pipe sizes over a threshold, resulting in higher pressure drops than anticipated by the Hagen-Poiseuille equation.

Poiseuille's equation addresses the pressure drop caused by a fluid's viscosity; other sorts of pressure drops may still exist in a fluid [For example, the pressure required to move a viscous fluid up against gravity would include both Poiseuille's law and Bernoulli's equation, such that each point in the flow would have a velocity greater than zero (otherwise no flow would happen).

Another example is that when blood flows into a tighter constriction, its speed is faster than in a wider diameter (because to continuity of volumetric flow rate), but its pressure is lower (according to Bernoulli's equation). The viscosity of blood, on the other hand, causes an extra pressure decrease along the direction of flow that is proportionate to the length travelled (as per Poiseuille's law). Both of these factors contribute to the pressure decrease.

Normally, Hagen-Poiseuille flow means not only the above-mentioned pressure drop relation, but also the complete solution for the parabolic laminar flow profile. However, the pressure drop finding may be extended to turbulent flow by inferring an effective turbulent viscosity, even if the flow profile in turbulent flow is not strictly speaking parabolic. The pressure drop in both circumstances, laminar and turbulent, is connected to the tension at the wall, which defines the so-called friction factor. The Darcy-Weisbach equation in hydraulics may be used to compute the wall stress phenomenologically, given a ratio for the friction factor in terms of the Prandtl number. In the case of laminar flow, consider the following for a circular cross section.

where Re is the Reynolds number, is the fluid density, and v is the mean flow velocity, which in the case of laminar flow is half the peak flow velocity. It is more beneficial to define the Reynolds number in terms of the mean flow velocity since this quantity stays clearly defined even in turbulent flow, while the maximum flow velocity may not be, or may be difficult to deduce in any case. In this version, the law approximates the Darcy friction factor, the energy (heads) loss factor, the friction loss factor, or the Darcy (friction) factor in laminar flow in a cylindrical tube at extremely low velocities. Wiedman separately deduced a slightly variant version of the law in

1856, and Neumann and E. Hagenbach independently in 1858. (1859, 1860). Poiseuille's law was named for the first time by Hagenbach.

Darcy’s Law

Darcy's law is an equation that explains the movement of fluids through a porous media. Henry Darcy developed the rule under discussion here based on the findings of research on the flow of water through sand beds, which constitute the foundation of hydrogeology, a branch of earth sciences.

Darcy's law was initially found empirically, although it has subsequently been derived from the Navier-Stokes equations through homogenization approaches. Ohm's law in the realm of electrical networks is equivalent to Fourier's law in the field of heat conduction.

The derivation of Darcy's law is widely utilised in petroleum engineering. It calculates the flow through permeable media, the most basic of which is a one-dimensional homogenous rock formation with a single phase fluid and constant fluid viscosity. Almost all oil reserves have a water zone under the oil leg, while some have a gas cap above the oil leg.

When reservoir pressure falls owing to oil production, water flows into the oil zone from below and gas flows into the oil zone from above if a gas cap exists, resulting in a flow of simultaneous and immiscible mixing of all fluid phases in the oil zone.

The operator in this context refers to the oil field, which may also inject water and/or gas to boost oil output. The flow of multiphase in oil and gas reservoirs is a broad issue, and one of many publications on the subject is the law of Darcy's for multiphase flow

Darcy's Law Explained in Depth

In geology, this is the law that explains the rate at which a fluid flows through a porous material. Darcy's law indicates that this rate is directly related to the decrease in vertical elevation between two points in the medium and indirectly proportional to the distance between them. The rule is used to explain the movement of water from one portion of an aquifer to another, as well as the flow of petroleum through the gravel and sandstone.

Darcy's law applies to laminar flow through sediments. Because the diameters of interstices in fine-grained material are minimal, the flow is laminar. Coarse-grained sediments act similarly, however the flow may be turbulent in extremely coarse-grained sediments. As a result, Darcy's law is not always applicable in such sediments.

For flow-through commercial pipes that are circular, the flow is laminar when the Reynolds number is less than 2000 and turbulent when it is more than 4000, while in certain sediments, the flow is laminar when the Reynolds number is less than 1.

For a very short time, the scales at a time that is derivative of flux may be added to Darcy's law, resulting in valid solutions at very tiny times that are in heat transfer. This is known as the modified version of Fourier's law.

The primary reason for doing so is because the ordinary groundwater flow equation is a diffusion equation, which leads to singularities at constant head limits at extremely short periods.

This is a more theoretically rigorous version, but it leads to groundwater, which is a hyperbolic flow equation that is more difficult to solve and is often out of the range of practical applicability.

Darcy's Law in Action

One use of Darcy's laws is in the study of water flow through an aquifer, which is the law of Darcy's which, along with the equation of conservation of that mass, reduces to the groundwater equation flow. Which is one of the fundamental hydrogeological interactions.

Muskat Morris was the first to modify Darcy's equation for a single-phase flow by incorporating viscosity in the single that is Darcy's fluid phase equation. This modification makes it appropriate for researchers in the petroleum business. Muskat and others' generalised flow, which is multiphase equations, offers the analytical basis for engineering, which is a reservoir that still exists today.

Darcy's Law Characteristics

Darcy's law is a simple mathematical statement that elegantly encapsulates many or we may say many common features of flowing groundwater in aquifers, which include: Because these are hydrostatic circumstances, if there is no pressure gradient across a distance, no flow occurs. Furthermore, if there is a pressure gradient, flow will proceed from high pressure to low pressure in the opposite direction of the growing gradient, hence the negative sign in Darcy's law.

Another factor is that the higher the pressure gradient that runs through the same formation material, the higher the discharge rate.

Darcy's law says that the flow of a fluid through a pipe or other closed conduit is proportional to the transverse force exerted on the fluid and inversely proportional to the fluid's viscosity. Lord Kelvin, a Scottish engineer, invented it in 1890. Lord Rayleigh conducted the first experimental demonstration in 1900.

In physics, Darcy's Law defines an imbalance in the permeability of porous materials. Lord Kelvin discovered it in 1856. Darcy's Law may be thought of as a description of how flow velocity in the pores of a porous media is governed by applied pressure and fluid characteristics. Another way to look at it is as a statement about how the size and geometry of the pore structure affect the permeability of the substance. The latter is the more common and well-known interpretation of Darcy's Law.

The equation is often referred to as a generalisation of Darcy's law. The pressure drops through the medium as the flow rises. The permeability of the material dictates how much fluid may flow in a given region, and the result of this flow is a reduction in fluid pressure. The formula is as follows: Q = flow rate, p = pressure drop, and L = pipe length. Here, K represents the permeability, is the Darcy velocity, and v represents the fluid viscosity. The equation holds true whether the system is static (no flow) or flowin

Continuity Equation

The mass flow equation of continuity (mass/volume per unit time) may be expressed as

where is the density, velocity, and volume per unit time.

The equation may be stated in terms of Darcy's Law by and is applicable for Newtonian fluids as well as non-Newtonian fluids such as gelatin or particle suspensions.

Darcy's Law formulation

Darcy's law may be broken down into two components: flow rate and pressure decrease. The hydraulic conductivity, in general, determines the flow rate, but it also influences the viscosity. As a result, the resistance to flow in a porous medium with varying permeability (e.g., porous rock, porous sand, porous concrete,...) is not always constant and may vary depending on the porous media structure or fluid viscosity (e.g. water in a rock). The pressure drop is provided. where P denotes the pressure drop, fluid velocity, permeability, and pipe length.

Darcy's law is related to fluid viscosity and length in this equation. The equation may be be generalised by include the porous medium resistance, as stated by Darcy's law. This indicates that the resistance of a porous material may change over time and space, for example as a consequence of flow erosion.

History

Lord Kelvin was researching the pressure distribution of a fluid passing through a tube. On June 29, 1890, he announced the law in a presentation to the British Association for the Advancement of Science in Edinburgh:

Kelvin was introduced to this equation without any formal rationale, although he had previously applied comparable equations to other fields of science, such as sound and water waves. In 1900, Lord Rayleigh presented the first experimental demonstration of this rule. Lord Rayleigh presented the first experimental evidence in his book, Fluid mechanics.

Since then, the equation has been used in a variety of scientific fields, including fluid dynamics, seismology, geophysics, nanoscale physics, biology, and materials science. J.D. Fardin, R. C. Desai, and S. M. Wise published the mathematical version of Darcy's law in 1975. T. M. Truskett has recently offered a comprehensive mathematical derivation of the equation.

Darcy's Law is derived

Darcy's Law is a simplified version of the more sophisticated Navier-Stokes equation. Lord Kelvin introduced it in 1890 as an alternate way to the Reynolds Stress Decomposition. Darcy's law has been developed using a variety of approaches, the first of which is a study published in 1975 by J.D. Fardin, R. C. Desai, and S. M. Wise. The derivation of an approximation Navier-Stokes equation for a simple pipe flow is discussed in this study.

According to Darcy's law, the flow of liquid is proportional to the pressure differential between two locations in a pipe. The two points are as follows:

1. A point A, which is placed at the pipe's inlet, and
2. A point B in the middle of the pipe that is also half the length of the pipe from A.

Two parameters may be used to indicate the flow of liquid via the pipe. They are the pipe's cross-section area and its kinematic viscosity. The pipe's cross-section area remains constant throughout its length. The flow rate affects the kinematic viscosity. Darcy's law is so named because Lord Kelvin developed it using the fluid mechanics theory.

Darcy's law in its precise version, where:

1. The liquid flow rate (liters per second)
2. The liquid's velocity (m/s)
3. The liquid's pressure (N/m2)
4. Kinematic viscosity (in m2/s)
5. the Darcy velocity (m/s) is the actual flow velocity.

The given equation is correct for constant, laminar flow. By inserting Darcy's law into the generic formula for constant laminar flow in a pipe, the Darcy velocity is produced. The equation yields the expressions for the terms.

Hydrostatic Force on a Submerged Surface

The resulting force acting on a submerged surface owing to the pressure loading of the fluid into which the surface is immersed is known as hydrostatic force. The centre of pressure is where the hydrostatic force acts.

Forces of Hydrostatics

As previously stated, hydrostatic force is the force acting on a submerged surface as a consequence of the fluid's pressure loading. When the liquid is at rest, there is no tangential force acting on the submerged surface, therefore the total pressure acts normally on the surface in contact. As a consequence, hydrostatic force is the force that acts on a submerged surface when the liquid is at rest.

Pressure Center

The centre of pressure is the point of the hydrostatic force. The centre of pressure is always lower than the centre of gravity of the surface with which it is in contact.

Formula and Calculation of Hydrostatic Force

1st Case: Horizontal Plane

The submerged surface is parallel to the water's surface in this scenario. The hydrostatic force may be computed easily as the product of fluid pressure and surface area (Figure 10.2).

The horizontal hydrostatic force,

$Fh = (\gamma * h) * A$

where,

- the liquid's unit weight

h - the depth at which the submerged aircraft may be found

A - the submerged plane's surface area

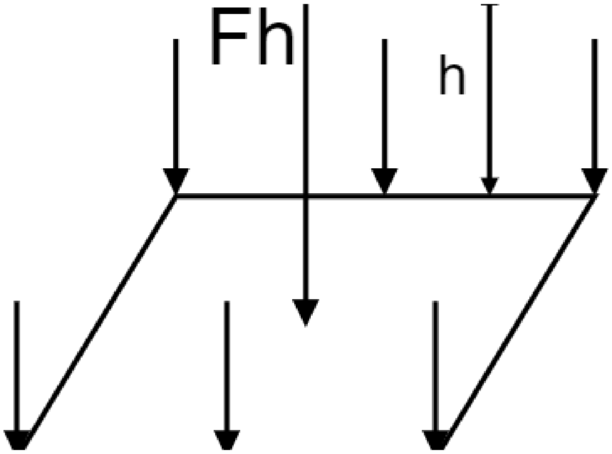

Figure 10.2 Hydrostatic Force on a Submerged Surface

Vertical plane

In this case, the submerged surface is vertical i.e., perpendicular to the surface of the water. Integration is required to derive the hydrostatic force on a vertical plane formula.

HydroStatic force on a vertical plane,

Fv = γ*A*h'

where,

γ - the unit weight of the fluid

A – total surface area

h' - the center of gravity of the surface

The center of pressure is given as,

hcp = h' + (Ig/Ah')

where,

Ig - the moment of inertia of the surface about its center of gravity

A - total surface area

h' - the center of gravity of the surface

The center of pressure (hcp) is always greater than the center of gravity (h') i.e., hcp > h'. Therefore, the center of pressure always lies below the center of gravity.

Curved Profile Surface

Unlike the simple surfaces we saw previously, the submerged surface in this instance has a curved contour. A sluice gate is an example of such a surface. In these circumstances, two forces, horizontal and vertical, are examined, and the resulting force is determined.

Horizontal force is calculated by taking into account the vertical surface, which is the vertical translation of the bent profile.

The horizontal force is denoted as

Fh = γ * A * h'

where,

- the liquid's unit weight

A is the area of the profile's vertical projection.

h' - the surface's centre of gravity

The weight of the liquid operating on the curved person in connection is referred to as vertical force.

The vertical force is denoted as

l = * Volume * Fv = * A contact

where,

- the liquid's weight in units

A contact - the profile's contact information

l - the profile's length

R = ((Fh2) + (Fv2))(1/2) Resultant force

Dimensional Homogeneity

Dimensional homogeneity states that in an equation, the sizes of each term on both sides are identical, which means that the dimensions of an equation's left-hand side (LHS) are comparable to the dimensions of its right-hand side (RHS) (RHS). The powers of basic dimensions on both sides of a dimensionally homogenous equation will be similar.

Dimensional Evaluation

Dimensional analysis is a mathematical approach used in research to assess model assumptions. Dimensional analysis is concerned with a physical quantity involved in the study. Quantities' dimensions are determined by their units. Quantities are classified into two types:

Fundamental quantities are quantities that are independent and invariant of any other quantity. They have a set size. Longitude, mass, time, temperature, electric current, luminous intensity, and quantity of matter are the seven basic quantities. L, M, T, A, I, and N are the dimensions of the seven basic quantities.

Derivative or subsidiary quantities are those that have more than one basic quantity and are reliant on the fundamental quantities. The force, for example, has the dimensions MLT2. As a result, force is a derived quantity.

Dimensions of two separate variables might be comparable; for example, acceleration and angular acceleration both have dimensions of LT2. Both velocity and angular velocity have dimensions of LT1. When the dimensions given to variables on both sides of an equation are equal, this is referred to as dimensional homogeneity. The dimensions are determined using quantity units. Dimensional homogeneity may be used to derive formulae for different physical quantities. Let us illustrate this with a few examples-

Example 1: The flow velocity is denoted by v = 2.g.h.

where v is the flow velocity, g represents the gravitational constant, and h is the flow head.

The velocity unit is ms1. As a result, the velocity dimensions are LT1.

ms2 is the unit of g. As a result, the dimensions of g are LT2.

The unit of h is the metre. As a result, the dimensions of h are L.

To ensure that the dimensions are homogeneous,

LHS dimensions = velocity dimensions = LT1

RHS Dimensions = 2.g.h Dimensions = (LT2. L)12 Dimensions = LT1 Dimensions

As a result, dimensional homogeneity occurs in the preceding equation.

Example 2: According to the kinematic equation, v = u + a.t

where u is the beginning velocity, a denotes the acceleration in metres per second squared, t denotes the duration, and v denotes the end velocity.

LHS dimensions = velocity dimensions =LT1

RHS dimensions = u + a.t dimensions = LT1 + (LT2).

(T) \s = LT−1

As a result, dimensional homogeneity occurs in the preceding kinematic equation.

Dimensional Analysis Methods Using Dimensional Homogeneity

If an equation has more than one variable, the relationship between the variables may be obtained using one of two methods:

Rayleigh's formula

Buckingham's approach

Rayleigh's formula

Rayleigh's approach is used to compute expressions for variables that are dependent on a maximum of four independent variables. If X is a variable that is reliant on X1, X2, and X3, then X is a function of X1, X2, and X3. Mathematically, the equation may be written as X = f. (X1, X2, X3).

X = k.Xa1. Xb2. Xc3 where a, b, and c are arbitrary powers and k is an additive term known as the non-dimensional constant.

Example

The equation for drag force on a sphere of diameter D moving with velocity v in a fluid of density and dynamic viscosity must be found, with the assumption that the equation will be dimensionally homogeneous. The following is a step-by-step solution.

As a result, the drag force F = k.Da.vb.c.d

Dimensionally, the LHS and RHS are identical. As a result, we shall equal the dimensions of variables on both sides.

Newton is the unit of force, and its dimension is MLT2.

The density unit is kg/m3, and the dimension is ML3.

The dynamic viscosity unit is Ns/m2, and its size is ML1T1.

As a result, MLT2 = k. La. (LT1).

b. (ML3)c. (ML1T1)d By equating the powers of M, L, and T on the left and right sides,

We get 1 = c+d.

1 = a+b-3c-d

and

-2 = -b-d, and so forth.

The equation is found by substituting all of the values in terms of d:

F = k. D2d. v2d. 1d. F

As a result, F = k..D2.v2.(.v.d)d

As a result, F = k..D2.v2. (.v.d)

where represents the function and k is the dimensionless additive term. The needed equation is given above.

Buckingham's approach

When there are more than four dependent or independent variables in an equation, Buckingham's approach is applied. According to this theorem, if a physical quantity has 'n' variables and these variables have'm' basic dimensions, then the variables are ordered in (n-m) dimensionless terms. If X1, X2,..., Xn are the variables and X1 is the dependent variable, and X2, X3,..., Xn are the independent variables on which X1 relies, then X1 = f (X2, X3, ..., Xn). The equation is sometimes written as f1 (X1, X2, X3,..., Xn). Buckingham's approach is sometimes known as Buckingham's pi-method since all (n-m) words are replaced with pi-terms.

Capillarity

Capillarity is the rise or descent of a liquid in a narrow route, such as a tube with a small cross-sectional area, such as the gaps between towel fibres or pores in a porous substance. Capillarity does not just exist in the vertical direction. Water is pulled into the strands of a towel regardless of how it is orientated.

Liquids rising in small-bore tubes put into the liquid are said to wet the tube, whilst liquids depressed inside thin tubes below the surface of surrounding liquid are said not to wet the tube. Water is a liquid than wets glass capillary tubes whereas mercury does not. Capillarity does not exist when there is no wetness.

Surface, or interfacial, forces cause capillarity. Water rises in a thin tube immersed in water due to forces of attraction between the molecule of water and the glass walls, as well as between the molecules of water themselves. These attractive forces simply balance the gravitational force of the column of water that has risen to a certain height. The higher the water rises, the smaller the diameter of the capillary tube. Mercury, on the other hand, is depressed to a larger extent as the bore narrows.

Vapour Pressure and Cavitation

When a liquid is drawn into a duct, it causes a pressure drop. If the fluid pressure falls below the saturation vapour pressure, the liquid starts to boil. This phenomena is known as cavitation in hydraulic (steam generation). The development of vapour bubbles as a result of pressure loss is known as cavitation.

At a given temperature, the saturation vapour pressure is the pressure at which a fluid transitions from the gaseous to the liquid state (or from liquid to gas). The pressure at which the fluid transforms from liquid to gas (saturated vapour pressure) increases as the temperature of the fluid rises. Thus, a liquid, such as water, may be converted to vapour at ambient pressure by applying heat; however, this transformation can be accomplished without altering the temperature by reducing the pressure gradient below the saturated vapour pressure. When a liquid is drawn into a duct, it causes a pressure drop. If the fluid pressure falls below the saturation vapour pressure, the liquid starts to boil. This phenomena is known as cavitation in hydraulic (steam generation). The development of vapour bubbles as a result of pressure loss is known as cavitation. The formation of bubbles increases the amount of fluid in the low differential pressure, which raises pressure in certain locations where the gas bubble condenses forcefully, bursting. The shocks produced by bursting bubbles damage the organ walls that come into touch with the fluid. A cavitation pump wears out rapidly.

$$\ln \frac{P_{sat}}{P_0} = \frac{M.L_v}{R}(\frac{1}{T_0} - \frac{1}{T})$$

Hydrostatic forces

Hydrostatic forces are the consequence of a liquid's pressure loading acting on buried surfaces. Fluid mechanics begins with the calculation of the hydrostatic force as well as the position of the centre of pressure.

Inclined and Curved Surfaces

When an item is put on a slanted surface, it often slides down the surface. The velocity at which the item slides down the surface is determined by its tilt; the higher the tilt of the ground, the quicker the object will slide down it. An inclined plane is a physics term for a slanted surface. An imbalanced force causes objects to accelerate down sloped planes. To comprehend this form of motion, consider the forces operating on an item on an inclined plane. The right-hand figure displays the two forces acting on a box positioned on a plane surface (assumed to be friction-free). As seen in the picture, every item positioned on a plane surface is always subject to at least two forces: gravity and normal force.

The Abnormal Normal Force

The primary feature of inclined plane issues is that the force f is not applied in the usual direction. We have always observed normal forces working in an upward direction, against the direction of gravity up to this point in the course. This is due to the fact that the items were always on sloping ground and never on inclined planes. The reality about normal forces is that they are always directed toward the surface that the item is on, not necessarily upwards.

The Components of the Gravity Force

Because the two (or more) impulses are not oriented in opposing directions, finding the net force exerted on an item on an inclined plane is a tough operation. As a result, one (or more) of the forces must be resolved into perpendicular components that can readily be added to the other forces operating on the object. Any motion controller at an angle to the horizontal is usually decomposed into horizontal and vertical elements. However, this is not the approach we shall use with inclined planes. Instead, the weight vector (Fgrav) will be divided into two perpendicular components in order to analyse the forces acting on objects on inclined planes. This is the second distinguishing feature of inclined plane issues. The force of gravity will be divided into two components, one parallel to the inclined surface and one perpendicular to the incline plane. The figure below depicts how gravity has been replaced by two components of force: a parallel and a perpendicular element of force.

The perpendicular component of gravity's force is directed towards the normal force and thereby balances it. There is no other force that can balance the parallel component of gravity. Due to the existence of an imbalanced force, this item will then accelerate down the airfoil. This acceleration is caused by the parallel component of gravity's force. The net force is the perpendicular component of gravity's force. The issue becomes significantly more difficult in the presence of friction or other forces (applied force, tectonic forces, etc.). Take a look at the diagram on the right. Because objects do not accelerate perpendicular to the slope, the perpendicular component of force still matches the normal force. However, while calculating the net force, the frictional force must also be included.

The net force, like in other net force issues, is the vector of all the forces. In other words, all of the separate forces are totaled together as vectors. The sum of the perpendicular component and the normal force is 0 N. The parallel element and the friction force add up to 5 N. The net force is 5 N, directed towards the floor down the slope.

The aforementioned issue (and other inclined plane problems) may be reduced by using a technique known as "tilting the head." An inclined plane issue is identical to any other net force question except that the surface has been angled. Simply tilt your head in the same way as the inclination was slanted to turn the issue back into the shape with that you feel more comfortable.

The issue should appear fairly familiar after the gravity's force has been resolved into its two components and the sloping plane has been slanted. Simply disregard gravity's force (which has been superseded by its two components) and solve for net force and acceleration. Consider the scenario represented in the figure to the right. The forces operating on a 100-kg container sliding down an air track are shown in the free-body diagram. The plane is sloped at a 30 degree angle. The friction coefficient between the container and the hill is 0.3. Determine the crate's net force and acceleration.

Begin by determining the force of gravity acting on the container, as well as the components of this force horizontal and perpendicular to the slope. Gravity's force is 980 N, and its components are Fparallel = 490 N (980 N • sin 30 degrees) and Fperpendicular = 849 N (980 N • cos30 degrees). The normal force may now be calculated to be 849 N. (it must balance the perpendicular component of the weight vector). The friction force may be calculated using the normal force and the coefficient of friction; Ffrict is 255 N (Ffrict = "mu"*Fnorm= 0.3 • 849 N). The vector sum of all forces is the net force. The forces perpendicular to the inclination balance; the forces parallel to the incline do not balance. The total net force is 235 N. (490 N - 255 N). 2.35 m/s/s (Fnet/m = 235 N/100 kg) is the acceleration.

Some Roller Coaster Physics

Roller coasters provide two thrills, the first of which is linked with the first plunge down a steep slope. The thrill of acceleration is provided by employing steep inclination angles on the initial descent; such big angles enhance the value of the weight vector's parallel component (the component that causes acceleration). Weightlessness is achieved by lowering the magnitude of normal force to levels smaller than their typical values. It is critical to understand that the sensation of weightlessness is connected with a lower than typical normal force. When sitting on a chair, a person weight 700 N will typically feel a 700 N normal force. If, on the other hand, the chair is moving down a 60-degree slope, the user will feel a 350 Newton normal force. This number is lower than usual, which leads to the sensation of weighing much less one's normal weight, i.e. weightlessness.

Archimedes’ Principle

Archimedes' principle is a physical law of buoyancy discovered by the ancient Greek mathematician and inventor Archimedes, which states that anybody whole or submerged in a fluid (gas or liquid) at rest is acted by a upward, or buoyant, force whose magnitude is equal to the weight of the fluid displaced by the body. The volume of displaced fluid is equal to the volume of a completely immersed item in a fluid or to a proportion of the volume just below surface of a partly submerged object in a liquid. The weight of the displaced fluid is proportional to the size of the buoyant force. The buoyant force applied to a body floating in a liquid or gas is similarly comparable in size to the floating item's weight and is directed in the opposite direction; the object does not rise or sink. A ship, for example, sinks into the ocean until the weight of the liquid it displaces equals its own weight. As the ship is loaded, it lowers deeper, displacing more water, and the magnitude of the buoyant force continues to match the ship's and cargo's weight.

If an item's weight is less than that of the displaced liquid, the piece rises, as in the instance of a piece of wood dropped under the surface of water or a helium-filled balloon discharged into the air. Though an item heavier than the quantity of fluid expelled sinks when released, the apparent weight loss is equal to the weight of the fluid displaced. In reality, some precise weighings need an adjustment to account for the buoyancy impact of the surrounding atmosphere. Assume a basin is completely filled with water. What happens to the level of water when an aluminium foil barge is put on it? The barge is claimed to be buoyant at this stage. But what would happen if many coins were placed to the barge? Will the barge remain afloat, or will it sink? In this case, the barge employs the idea of buoyancy. What exactly does the term "buoyant" mean? What is the definition of buoyant force? In the next sections, you will learn more about buoyancy definition, buoyant force, and its applications.

Buoyancy is the propensity of an item to float in any fluid. The buoyant force keeps objects floating in fluids. It is an upward force that the liquid exerts on the submerged object. Buoyant force is shown by a ship floating in the middle of the sea, an anchor sinking when dropped in the water, and even a fish hovering in the centre.

The idea of buoyancy was developed by Archimedes (287-212 B.C.), a Greek mathematician, scientist, and astronomer. His famous Eureka! Moment came when he was sitting in a tub, contemplating how to verify whether King Hieron II of Syracuse's crown is really composed of gold or has been mingled with silver. He then solved the problem by comparing the amount of water displaced by the king's crown to the volume displaced by a bar of gold of the same mass. He noticed that the crown had been tampered with and that the jeweller had really defrauded the monarch.

Nozzle and Nozzle Wear

Nozzle

A nozzle is always a pipe of tube with a variable cross sectional area that is used to control, direct, or alter the flow of liquid (liquid or gas). Nozzles are widely employed to regulate the stream that flows from them in terms of its mass, shape, speed, direction, pressure, and/or rate of flow. The melt can exit the barrel and enter to mound through the nozzle, which is fitted into the barrel of the gun. Additionally, it is a location where the melt may be warmed by friction and conduction before entering the mold's relatively cold channel. When the nozzle comes into contact with the mold, heat is transferred from the nozzle; however, if this heat transfer is considerable, it is best to remove the nozzle first from mold as during screw-back phase of a molding cycle. If not, the plastic might freeze inside the nozzle

Nozzles are contoured conduits used to accelerate liquids or gases to a desired speed in a predetermined direction. Nozzles are employed in gas dynamic lasers, gas turbines, jet devices, intense shattering or spraying technologies, rocket and aviation engineering to create jet propulsion (see Gas turbine). The fundamental building blocks of wind tunnels (see Wind Tunnels) are nozzles, which enable the construction of systems for intense (jet) cooling, like in electrical engineering

Types of Nozzle

The types of nozzle is described into four categories:

1. Jets nozzles
2. High velocity nozzles
3. Propelling nozzles
4. Spray nozzles

Jets Nozzles

A nozzle used to propel gas or liquid into the environment in a cohesive stream is known as a jet, fluids jet, or hydro jet. Gas jets are frequently seen in gas stoves, ovens, or grills. Prior to the invention of electricity, gas jets were a prominent source of illumination. In carburetors, where smooth, calibrated orifices are utilized to control the flow of gasoline into an engine, there are additional varieties of fluid jets

High velocity nozzles

For usage in fixed wash water or inundation systems for fire prevention applications, high velocity water sprays nozzles are external swirl plate kind open nozzles. To put out fires, these nozzles provide a solid, consistent, and dense center of high-velocity water spray.

Propelling nozzles

A variable area propelling nozzle through its open configuration is used by engines that need to generate thrust fast, from idle, in order to minimize thrust and maintain high engine rpm. A propelling nozzle is a part of a jet engine that is located at the tail of something like the engine and operates to form a combustion jet and to maximize the use of the exhaust stream energy throughout the engine. Closing the nozzle toward the powerful position is easy and quick when propulsion is required, such as when starting a go-around.

Spray nozzles

A basic tool called a spray nozzle is used to divide a water dynamics into a spray nozzle. Despite the nozzles' apparent simplicity, there are many distinct items in our assortment that represent the many ways that diverse sectors require to spray different fluids.

Nozzle Wear

The nozzle's wear mechanism is comparable to those of other conventional machining operations that employ cutting tools. The nozzle wall may deteriorate due to the water's high pressure and harsh abrasive particles. Fiber reinforced polymer is a material that may be used with 3D printers more and more often. Carbon fiber is the most used type of filler. With more people using 3D printing, fiberglass is moving up to second. Long before 3D printing, it was usual in injection molding. These extra fibers are intended to improve the mechanical qualities. In general, see a rise in bending modulus and tensile strength. These polymer and fiber blends are referred to as "Filled Plastics" and "Composites" at times.

Nozzle wear falls into one of two categories:

Breakage of nozzle body

The nozzle's spray pattern, pressure, and flow will all be subpar. Mechanical harm caused accidentally. The most frequent issue resulting in subpar spray nozzle performance and nozzle replacement is this one. Poor spray patterns, higher pressure, and lower flow are the results. The throat Mach number cannot rise over one even after boosting the nozzle pressure ratio further. The flow is ready to develop to supersonic speeds downstream (i.e. outside the nozzle), however Mach 1 can be a high speed for a hot gas since the speed of sound changes with the squared of absolute temperature. This knowledge is often applied in the rocketry industry, where hypersonic velocities are necessary and certain propellant combinations are utilized to further boost sonic speed .Divergent nozzles speed up sonic or hypersonic fluids while slowing subsonic fluids. After making contact with the coal's surface, the minute water droplets will begin to move laterally under a specific amount of pressure.

Water is driven to enter the pores and fractures of the coal body then interact with them because of the pressure differential between the liquid water and the hole fluid on the inside of the coal species, which is similar to imparting a quasi-static pressure on the coal body. When the contact reaches critical force for fracture propagation, the pores and cracks would keep growing and converge to generate a secondary failure. Continuous high-pressure liquid passing through the spray gun opening over time is what causes erosion. The flow widens the aperture while progressively removing metal. The outcome is an uneven spray pattern, an increase in flow, a drop in pressure.

Turbine and its concept

Turbine

Turbine is a machine which takes the angular momentum of fluids and turns into mechanical energy throughout a specified mechanism. The turbine consist of many blades which are attached to an axle and is used typically to operate a generator. A turbine is a rotating mechanical device that takes energy from the fluid flow and transforms it into productive work (from the Greek, tyre, or Spanish turbo, meaning vortex). When used in conjunction with a generator, the work that a turbine does may be utilized to create electricity. A turbine is a components of performance having at least single moving component called a rotor component, which is a shafts of drum with connected blades. The blades are affected by a moving fluid, which causes them to move and give the rotor rotational energy. Waterwheels and windmills are two early turbine types

Hydraulic Machines

The term "hydraulic machines" refers to any device that transforms hydraulic energy (the energy contained in water) into mechanical energy, which is then transformed into electrical energy, or mechanical power into hydraulic energy.

Turbines are hydraulic devices that transform hydrodynamic energy into mechanical energy. By definition, turbines are hydraulic devices that transform hydraulic energy into mechanical energy. An electric generator that is directly connected to the turbine shaft uses this mechanical energy to run. This results in the conversion of mechanical energy to electrical energy. Hydro-electric power is the name given to the electric power produced by hydraulic energy (water energy)

Types of Turbine

The turbine are of four types

1. Steam Turbine
2. Vapor Turbine
3. Gas Turbine
4. Hydraulic Turbine

Hydraulic Turbine

A rotating device known as a hydraulic turbine or turbine shaft uses the potential and kinetic energy of moving water to produce mechanical work. The primary device that aids in converting hydrostatic energy into electricity with the aid of a generator is a hydraulic turbine. The turbine is forced to revolve when a jet of water strikes its blades; thus, the turbine is equipped with a generator, which likewise rotates and generates electrical energy

Vapor Turbine

A steam turbine is a device that uses pressurized steam's thermal energy to drive mechanical energy on a revolving output shaft. Charles Parsons created it in its current form in 1884. Modern steam turbines are made utilizing sophisticated metalworking techniques that were first made possible in the 20th century. The continuing improvement of steam turbines' resilience and efficacy is still essential to the economics and finance of the 21st century.

Gas Turbine

A steam turbine operates by heating water to extreme temperatures until it is transformed to steam using a heating element (gas, coal, nuclear, or solar energy). That steam begins to cool as it passes through the turning turbine blades. Thus, in the rotating turbine's blades, the steam's potential energy is converted to kinetic energy. Steam turbines are particularly well adapted for operating electrical generators for the creation of electrical power since they produce rotational motion. The generator, which has an axle connecting it to the turbines, generates energy by creating a magnetic field that induces an electric current

Steam Turbine

A steam turbine is a device that uses pressurized steam's thermal energy to drive mechanical work on a revolving output shaft. Charles Parsons created the steam turbine as we know it today in 1884. Fabricating a modern steam turbine requires advanced metalwork to shape high-grade steel alloys into precise parts using technologies who first became accessible in the 20th century; steam turbines' ongoing improvements in robustness and efficiency remain crucial toward the energy economics of the 21st century. The utilization of many stages in the expansions of the steam, and results in a review of current to the possible responses expansion process, is a major contributor to the steam turbine's gain in thermodynamic efficiency.

Application of Turbines

One of the tools that engineers utilize the most frequently today is the turbine. They are spinning devices with a variety of uses, spanning hydroelectric power to racing, as well as many purposes. Turbines are used in nuclear, hydro, thermoelectric, and other energy plant types. Wind turbines are also receiving even more investment and use in this industry. They are essentially motor machines since the working fluid moves them. For instance, the idea behind counter-working pumps. The working fluid is energized by pumps, which raises its pressure and encourages fluid flow in the pipeline.

Each of these models has a bladed rotor, which makes it distinct from other designs. Except perhaps wind turbines, it is built with nozzles, the fixed blades that are in charge of guiding the fluid toward the rotating blades. At this point, the fluid's enthalpy1 is converted into mechanical power, which causes an axis to rotate. This axis is connected to power generator in power generation turbines, which facilitates the production of electrical energy. The air that moves through the turbines used during aviation undergoes compression accompanied by combustion before exiting the device, creating the thrust required to operate an aero plane.

They are often operated with saturated steam with superheated steam inside of plants, where each application mostly maintains the machine's constructive structure. Turbine discharge pressure is a significant and often used classification. Backpressure turbines are turbines with discharge pressures higher than atmospheric. The turbine is referred to as condensation if the outlet pressure is reduced than pressure and temperature. Condensing turbines are typically big turbines that use superheated steam to function. The steam expands through multiple stages before condensing to produce just water at the conclusion of the operation.

Centrifugal Pump and its specification in Fluid Mechanics

Centrifugal Pump

Centrifugal pumps carry fluids by transferring rotational kinetic energy into water circuit flow. Rotational energy is often generated by a turbine or electric engine. The fluid passes the pump impeller along or towards the rotating axis and is driven either by turbine before flowing outwards into a distributor with volute compartments (casing) and exiting. They belong to the dynamic axial effort turbomachinery class. A centrifugal pump moves a fluid by transferring rotational momentum by one or more machine systems to one or perhaps more driven rotors known as impellers. Fluid is propelled out of the rapidly rotating propeller along its circle by centrifugal force through the vane tips. The pump output is achieved by the fluid going at a greater speed and pressure due to the functioning of the impeller. The flowing fluid is directed by the impeller into the pipe wall, to be confined, slowed, and regulated before being expelled.

Centrifugal pump working

The impeller of a centrifugal pump is its most significant component. It is composed of many bent vanes. They are usually situated between the two plates (an enclosed impeller). For water containing entrained materials, an open closed tractor trailers impeller should have been employed.

The impeller's axis (the "eye") receives fluid, which departs along the circle between the vanes. The impeller is attached to a motor and rotates quickly on the side opposite the eye by a driving shaft (typically 500-5000rpm). The fluid is accelerated through to the impeller vanes and into the suction port by the impeller's rotating motion. Pump casings come in two different fundamental types: volute and diffuser. Both designs' objectives are to convert fluid flow into a regulated discharge at pressure. The impeller is offset in a volute casing, essentially forming a curved vortex with a growing cross-sectional area as it approaches the pump outlet.

Questions for Revision

1. Define fluid behaviour and differentiate between fluid statics and fluid dynamics
2. What is fluid mechanics?
3. State fluid flow.
4. State fluid characterstics.
5. Describe types of fluids.
6. Describe Centrifugal Pump.
7. Describe Steam Turbine.
8. Define laminar and turbulent flow.
9. Describe Gas Turbine.
10. Describe Archimedes principle.
11. What are Hydraulic Machines? Descibe their working principles.
12. Describe nozzle and wear.
13. Describe notch and waer.
14. Describe Burnowli's priciples.
15. Define Hook's law and estimate eueler equation.

Reference related to further study

1. A Textbook of Fluid Mechanics and Hydraulic Machines by R.K. Bansal
2. Fluid Mechanics and Hydraulic Machines by McGraw Hill Education (India).
3. Fluid Mechanics by Yunus Cengel, Jhon Cimbala, Tata Macgraw Hill,New Delhi.
4. .Fluid Mechanics by R. J. Garde, A.J Mirajgaonkar, SCITECH Publication.
5. Fluid Mechanics by Streeter & Wylie, Tata McGraw Hill
6. Fluid Mechanics by Dr. A. K. Jain, Khanna Publishers.
7. Fluid Mechanics by K. Subramanya, McGraw Hill.s
8. Fluid Mechanics by Frank White, McGraw Hill.
9. . Fluid Mechanics- Fundamentals and Applications by Cengel and Cimbala- McGraw Hilll
10. Flow through Open Channels—Srivastava– Oxford University Press
11. A test book of Fluid mechanics and Machinery by Bansal
12. . Fluid Mechanics by Streeter, Wylie, and Bedford – Tata McGraw Hill

----