Theoretical approach of fluid dynamics in significant area

Rupen Chatterjee

Department of Mathematics, Nabagram Hiralal Paul College, Nabagram, Hooghly, West Bengal Pin:712246, India (Affiliated by Calcutta University)

ARTICLE DETAILS

Article History
Published Online: 20 January 2019

Keywords
Fluid dynamics, liquid, biological

ABSTRACT

Fluid dynamics has a very significant role. It takes little more than a brief look around for us to recognize that fluid dynamics is one of the most important of all areas of physics life as we know it would not exist without fluids, and without the behavior that fluids exhibit. The air we breathe and the water we drink and which makes up most of our body mass are fluids. Motion of air keeps us comfortable in a warm room, and air provides the oxygen we need to sustain life. Similarly, most of our (liquid) body fluids are water based. And proper motion of these fluids within our bodies, even down to the cellular level, is essential to good health. It is clear that fluids are completely necessary for the support of carbon-based life forms. But the study of biological systems is only one and a very recent one possible application of a knowledge of fluid dynamics. Fluids occur, and often dominate physical phenomena, on all macroscopic non-quantum length scales of the known universe from the megaparsecs of galactic structure down to the micro and even nanoscales of biological cell activity. In a more practical setting, we easily see that fluids greatly influence our comfort or lack thereof they are involved in our transportation systems in many ways they have an effect on our recreation. From this it is fairly easy to see that engineers must have at least a working knowledge of fluid behavior to accurately analyze many, if not most, of the systems they will encounter. It is the goal of these lecture notes to help students in this process of gaining an understanding of, and an appreciation for, fluid motion what can be done with it, what it might do to you, how to analyze and predict it.

1. Introduction

In this introductory we will begin by further stressing the importance of fluid dynamics by providing specific examples from both the pure sciences and from technology in which knowledge of this field is essential to an understanding of the physical phenomena (and, hence, the beginnings of a predictive capability—e.g., the weather) and/or the ability to design and control devices such as internal combustion engines. We then describe three main approaches to the study of fluid dynamics: i) theoretical, ii) experimental and iii) computational; and we note (and justify) that of these theory will be emphasized in the present lectures. We have already emphasized the overall importance of fluids in a general way, and here we will augment this with a number of specific examples. We somewhat arbitrarily classify these in two main categories: i) physical and natural science, and ii) technology. Clearly, the second of these is often of more interest to an engineering student, but in the modern era of emphasis on interdis ciplinary studies, the more scientific and mathematical aspects of fluid phenomena are becoming increasingly important.

Fluids in the pure sciences

The following list, which is by no means all inclusive, provides some examples of fluid phenomena often studied by physicists, astronomers, biologists and others who do not necessarily deal in the design and analysis of devices.

1. Atmospheric sciences (a) global circulation: long-range weather prediction; analysis of climate change (global warming) (b) mesoscale weather patterns: short-range weather prediction; tornado and hurricane warnings; pollutant transport

2. Oceanography (a) ocean circulation patterns: causes of El Ni˜no, effects of ocean currents on weather and climate (b) effects of pollution on living organisms

3. Geophysics (a) convection (thermally-driven fluid motion) in the Earth's mantle: understanding of plate tectonics, earthquakes, volcanoes (b) convection in Earth's molten core: production of the magnetic field

4. Astrophysics (a) galactic structure and clustering (b) stellar evolution—from formation by gravitational collapse to death as a supernovae, from which the basic elements are distributed throughout the universe, all via fluid motion

5. Biological sciences (a) circulatory and respiratory systems in animals (b) cellular processes

2. Fundamental Concepts of Fluid Dynamics

This chapter is devoted to the discussion and presentation of various concepts that are important to understand the heat and mass transfer in the boundary layer flow of nanofluids. We start this chapter with a review of the fundamental concepts of fluid mechanics that form the framework for heat and mass transfer in the nanofluid flow. We first present the basic equation of motion, heat and mass transfer. We then present the fundamentals of nanofluids and its applications. Finally, the review of the previous related studies is presented.

Fluid is a substance that continually deforms (flows) under an applied shear stress. Fluids are a subset of the phases of matter and include liquids, gases, plasmas and, to some extent, plastic solids. Liquids form a free surface (that is, a surface not created by the container) while gases do not. The gross properties of solids, liquids and gases are directly related to their molecular structure and to the nature of the forces between the molecules. The distinction between solids and fluid is not a sharp one, since there are many materials which in some respects behave like a solid and in the other respects like a fluid. The distinction between liquids and gases is much less fundamental, so far as dynamical studies are concerned. For reasons related to the nature of intermolecular forces, most substances can exist in either of two stable phases which exhibit the property of fluidity, or easy deformability. The density of a substance in the liquid phase is normally much larger than that in the gaseous phase, but this is not in itself a significant basis for distinction since it leads mainly to a difference in the magnitudes of forces required to produce given magnitudes of acceleration rather than to a difference in the types of motion. The most important difference between the mechanical properties of liquids and gases lies in their bulk elasticity, that is, in their compressibility. Gases can be compressed much more readily than liquids, and as a consequence any motion involving appreciable variations in pressure will be accompanied by much larger changes in specific volume in the case of a gas than in the case of a liquid.

Fluid mechanics is concerned with understanding, predicting, and controlling the behavior of a fluid. Since we live in a dense gas atmosphere on a planet mostly covered by liquid, a rudimentary grasp of fluid mechanics is part of everyday life. For an engineer, fluid mechanics is an important field of the applied sciences with many practical and exciting applications. If you examine municipal water, sewage, and electrical systems, you will notice a heavy dependence on fluid machinery. Pumps and steam turbines are obvious components of these systems, as are the valves and piping found in your home, under your city streets, in the Alaska oil pipeline, and in the natural gas pipelines that crisscross the country. Moreover, aircraft, automobiles, ships, spacecraft, and virtually all other vehicles involve interactions with fluid of one type or another, both externally and internally, within an engine or as part of a hydraulic control system. Learning more about fluid mechanics also allows us to better understand our bodies and many interesting features of our environment. The heart and lungs, for example, are wonderfully designed pumps that operate intermittently rather than steadily as most man-made pumps do. Yet the heart moves blood efficiently through the branching network of arteries, capillaries, and veins, and the lungs cycle air quite effectively through the branching pulmonary passages, thereby keeping the cells of our bodies alive and functioning. Many other sophisticated fluid handling devices are found throughout the biological world in living creatures of all types, sizes, and degree of complexity. The environment is another source of complex and interesting fluid mechanics problems. These range from the prediction of weather, hurricanes, and tornadoes to the spread and control of air and water pollution. Add to this list the flow of rivers and streams, the movement of groundwater, the jet stream and great ocean currents, and the tidal flows in estuaries. The lava flows of volcanoes and the movements of molten rock within the earth also lie within the domain of fluid mechanics. Looking beyond Earth, stellar processes and interstellar events are striking examples of fluids in motion on a grand scale. Knowledge of fluid mechanics is also the key to understanding and sometimes controlling other interesting, if not vital phenomena, such as the curving flight of a tennis, golf, or soccer ball, and the many different pitches in baseball.

The field of fluid mechanics has historically been divided into two branches, fluid statics and fluid dynamics. Fluid statics, or hydrostatics, is concerned with the behavior of a fluid at rest or nearly so. Fluid dynamics involves the study of a fluid in motion.

Modern engineering science is rooted in the ability to create and solve mathematical models of physical systems. Fluid mechanics is often considered as a challenging subject, primarily because the underlying mathematical model appears to be complex and difficult to apply. The governing equation of fluid statics, called the hydrostatic equation, is actually relatively simple and may always be solved to find the pressure distribution in the fluid. On the other hand, the governing equation of fluid dynamics, called the Navier-Stokes equation, would never be described as simple. The inherent difficulties of fluid mechanics have been recognized for centuries, yet engineers have demonstrated great ingenuity in developing a number of different approaches to solving specific fluid flow problems. The common theme is to simplify the mathematical or experimental model used to describe the flow without sacrificing the relevant physical phenomena. The art of fluid mechanics, however, is developed primarily through experience, both your own and that of others. This art consists in knowing when it is safe to neglect the effects of physical phenomena that are judged to have little impact on the flow. Once we decide to neglect certain physical phenomena, we drop the corresponding terms in the governing equations, thereby decreasing the difficulty in obtaining a solution. Such fundamental topics as boundary layer theory, the Bernoulli equation, potential flow, and even fluid statics can be considered to be part of the art of fluid mechanics in this sense. Learning about these historical approximations and others like them is essential in fluid mechanics, and engineers working with fluids should continue to seek to attain it.

Once the analysis of a fluid mechanics problem has been cast in the form of an appropriate mathematical model, a solution method must be chosen. For example, one might employ an analytical solution method that results in a representation of the flow variables as functions of space and time. An analytical solution is a highly compact and useful form of solution that should always be acquired if possible. Be aware, however, that an analytical solution of the governing equations of fluid dynamics is usually not possible. Complex engineering geometries and a natural tendency for fluid flows to become unstable ensure that analytical solutions will remain elusive. Nevertheless, it is wise to consult the engineering literature to determine what has been accomplished in treating the same or related flow problems. If you find that an approximate analytical solution to a problem of current interest is available, you may be able to use it as the starting point for your analysis. Today, an engineer will increasingly choose to employ computational methods to solve the equations of fluid motion. These methods include finite difference, finite element, finite volume, and other computational approaches in which digital computers are used to supply numerical solutions of approximate versions of the governing equations. These solutions are discrete, meaning that the flow variables are known only at specific spatial locations in the flow field. Computational tools of all kinds, ranging from commercially available computational fluid dynamic codes to visualization packages and symbolic mathematics codes, are among the most important aids in the modern practice of fluid mechanics. The one motivation in writing this thesis is to integrate some of the modern computational aids into fluid mechanics. The computer programming languages C, C++, Mathematica, Maple, Matlab, and others like them are superb aids in learning fluid mechanics. It is recommended using them to simplify calculations and to visualize the mathematics.

3. Methods and Analysis of Theoretical Approach of Fluid Dynamics with Some Significant Definition

This study will cover title of the study, significance of the study, aims and objectives of the study, research hypothesis and research design. This research has designed based upon descriptive study as it aims to identify and elaborate the objectives of work.

- The research design contains the following steps:
- Literature review

Theoretical and experimental analysis.

This study combines both primary and secondary research methods. Thus, gathering and analyzing the data will be done on the basis of existing research.

SPSS statistical package of data analysis will employ to analyze the quantitative data.

Secondary data

The secondary data has been collected from various journals, books and policy documents of the government.

Primary data

Stratified random sample technique has been followed to identify the respondents. A Structured Questionnaire was designed, tested and administered for collection of data.

Fluid dynamics is the science treating the study of fluids in motion. By the term fluid, we mean a substance that flows i.e. which is not a solid.

Air flow and heat transfer within a fluid are governed by the principles of conservation of mass, momentum, and thermal energy. As the equations describing the conservation laws are coupled set of second order partial differential equations, there is no analytical solution available. Therefore, in order to predict the airflow and temperature, at any given point in the isolation room space, CFD techniques are used. Further, while an Eulerian approach is required for CFD technology, particle tracking is more readily implemented using a Lagrangian approach. This section describes both the CFD technology, and the simulation of the bacteria particles.

Boundary layer

A very narrow region next to a solid object in a moving fluid, and containing high gradients in velocity.

Clustering

Increasing the number of grid points in a region to better resolve a geometric or flow feature. Increasing the local grid resolution.

Continuum

Having properties that vary continuously with position. The air in a room can be thought of as a continuum because any cube of air will behave much like any other chosen cube of air.

Convection

A similar term to Advection but is a more generic description of the Advection process.

Convergence

Convergence is achieved when the imbalances in the governing equations fall below an acceptably low level during the solution process.

Diffusion

The process by which a quantity spreads from one point to another due to the existence of a gradient in that variable.

Diffusion, molecular

The spreading of a quantity due to molecular interactions within the fluid.

Diffusion, turbulent

The spreading of a quantity due to the increased mixing rates exhibited by turbulent flows. In the majority of situations turbulent diffusion far exceeds molecular diffusion.

Divergence

Divergence occurs when the imbalances in the governing equations reach unacceptably high levels during the solution process.

Eddy viscosity

Eddy viscosity is an additional viscosity that is produced due to the effects of fluid turbulence.

Eddy diffusivity

Eddy diffusivity is the additional diffusivity produced due to the effects of fluid turbulence.

Far-field distance

The approximate distance from the surface of the body to the farthest point in the Computational Domain. "The wing simulation had a far-field distance of 15 wing chords."

Gauss Siedel equation solve

A method by which linear equations are solved on a cell-by-cell basis Gradient: The amount by which a variable changes in space or time.

Grid resolution

The amount of grid points located in a physical area. "The grid uses 20 grid points to resolve the boundary layer”. Near-wall spacing: The distance of the closest point to the surface of a body. An especially important parameter in viscous flow simulations.

Normal stress

The force/unit area that results from one body directly striking another. For instance, slamming your fist down upon a table top will cause pain due to a normal stress on your hand and the table. Pressure is always a normal stress.

Reynolds number

A non-dimensional number that is used to indicate how turbulent a fluid flow .

Reynolds stress

In turbulence modeling an instantaneous velocity is broken down into mean and fluctuating components. A Reynolds stress is the averaged product of two of these fluctuating velocity components.

Reynolds flux

A Reynolds flux, as in a Reynolds stress, is the average product of two fluctuating variable components, one of which is a fluctuating velocity component.

Shear stress

The force/unit area that results from one body sliding relative to another. For instance, sliding a book along a table top will cause a shearing stress on both the book and table top.

Solution domain

The computational volume in which the governing equations, together with the boundary conditions, are solved

Turbulence

Turbulence is a type of flow that occurs when a fluid is moving quickly and / or within an unconfined space. It is characterized by a marked increase in mixing where, superimposed on the principle motion, there are countless irregular fluctuations.

Viscosity

The viscosity of a fluid is ascribed to the movement of one layer of fluid over another, i.e. a viscous fluid like maple syrup will take a long time to pour from a bottle, while beer can be ". Water is about 100 timesμpoured quite readily. Viscosity is usually given the Greek symbol " as viscous as air, while most oils are around 1000 times as viscous as water. The effects of viscosity are most easily related to a concept like friction. The viscosity of fluids will cause a resistance to motion, a drag, which must be overcome by providing more power. If the drag caused by viscosity is small compared to other forces, or if it is important only in a small region like in Boundary Layer Theory, then the effects of viscosity can be neglected. Such a case is called inviscid flow. It is a point of confusion, even for practicing aeronautical engineers, that an inviscid flow is not the flow of a fluid with zero viscosity, rather an inviscid flow contains negligibly small viscous stresses.

Vorticity

Vorticity is the swirling motion of a fluid. Satellite photographs on the evening news weather forecast often show large rotating masses of fluid, which are special cases of vortices.

CFD

Computational Fluid Dynamics can be summarized by the following definitions:

Computational

The computational part of CFD means using computers to solve problems in fluid dynamics. This can be compared to the other main areas of fluid dynamics, such as theoretical and experimental.

Fluid

When most people hear the term fluid they think of a liquid such as water. In technical fields, fluid actually means anything that is not solid, so that both air and water are fluids. More precisely, any substance that cannot remain at rest under a sliding or shearing stress is regarded as a fluid.

Dynamics

Dynamics is the study of objects in motion and the forces involved. The field of fluid mechanics is similar to fluid dynamics, but usually is considered to be the motion through a fluid of constant density. This shows that CFD is the science of computing the motion of air, water, or any other gas or liquid.

Overview of CFD

The science of computational fluid dynamics is made up of many different disciplines from the fields of aeronautics, mathematics, and computer science. A scientist or engineer working in the CFD field is likely to be concerned with topics such as stability analysis, graphic design, and aerodynamic optimization. CFD may be structured into two parts, generating or creating a solution, and analyzing or visualizing the solution. Often the two parts overlap, and a solution is analyzed while it is in the process of being generated in order to ensure no mistakes have been made. This is often referred to as validating a CFD simulation.

CFD Solutions

When scientists or engineers try to solve problems using computational fluid dynamics, they usually have a specific outcome in mind. For instance, an engineer might want to find out the amount of lift a particular airfoil generates. In order to determine this lift, the engineer must create a CFD solution, or a simulation, for the space surrounding the airfoil. At every point in space around the airfoil, called the grid points, enough information must be known about the state of a fluid particle to determine exactly what direction it would travel and with what velocity. This information is called flow variables.

We begin by introducing the "intuitive notion" of what constitutes a fluid. As already indicated we are accustomed to being surrounded by fluids—both gases and liquids are fluids—and we deal with these in numerous forms on a daily basis. As a consequence, we have a fairly good intuition regarding what is, and is not, a fluid; in short we would probably simply say that a fluid is "anything that flows." This is actually a good practical view to take, most of the time. But we will later see that it leaves out some things that are fluids, and includes things that are not. So if we are to accurately analyze the behavior of fluids it will be necessary to have a more precise definition. This will be provided in the next chapter. It is interesting to note that the formal study of fluids began at least 500 hundred years ago with the work of Leonardo da Vinci, but obviously a basic practical understanding of the behavior of fluids was available much earlier, at least by the time of the ancient Egyptians; in fact, the homes of well-to-do Romans had flushing toilets not very different from those in modern 21st-Century houses, and the Roman aquaducts are still considered a tremendous engineering feat. Thus, already by the time of the Roman Empire enough practical information had been accumulated to permit quite sophisticated applications of fluid dynamics. The more modern understanding of fluid motion began several centuries ago with the work of L. Euler and the

Bernoullis (father and son), and the equation we know as Bernoulli's equation (although this equation was probably deduced by someone other than a Bernoulli). The equations we will derive and study in these lectures were introduced by Navier in the 1820s, and the complete system of equations representing essentially all fluid motions were given by Stokes in the 1840s. These are now known as the Navier–Stokes equations, and they are of crucial importance in fluid dynamics. For most of the 19th and 20th Centuries there were two approaches to the study of fluid motion: theoretical and experimental. Many contributions to our understanding of fluid behavior were made through the years by both of these methods. But today, because of the power of modern digital computers, there is yet a third way to study fluid dynamics: computational fluid dynamics, or CFD for short. In modern industrial practice CFD is used more for fluid flow analyses than either theory or experiment. Most of what can be done theoretically has already been done, and experiments are generally difficult and expensive. As computing costs have continued to decrease, CFD has moved to the forefront in engineering analysis of fluid flow, and any student planning to work in the thermal-fluid sciences in an industrial setting must have an understanding of the basic practices of CFD if he/she is to be successful. But it is also important to understand that in order to do CFD one must have a fundamental understanding of fluid flow itself, from both the theoretical, mathematical side and from the practical, sometimes experimental side.

4. Application and Results

Eulerian formulation: mass-momentum conservation

The general statement is very simple: Change per unit time = Flux-in - Flux-out The relevant quantities are mass-momentum-energy.

The mass in a cube of volume $V = \Delta x\ \Delta y\ \Delta z$ is $M = \rho V$ and the flux across the six faces of the cube has the form $J_i = \rho u_i A_i$, where $A_i = \Delta x_j \Delta x_k$, with i, j, k = x, y, z. The mass balance dM/dt = 0 applied to the cube of fluid in the limit of zero volume delivers the continutity equation:

$$\partial_t \rho + \partial_i(\rho u_i) = 0$$

The same argument applied to the momentum in the cube of fluid, $P_i = M u_i$, and taking into account the forces acting on the surfaces, deliver the NavierStokes equations.

$$\partial_t \rho u_i + \partial_j (\rho u_i u_j + P\delta_{ij}) = \partial_j \sigma_{ij} + F_i$$

where P is the pressure, σ_{ij} is the stress tensor and F_i the external force per unit volume.

Note that these forces are of two types: i) contact forces, due to the pressure/stress exerted by abutting faces of neighbor cubes, ii) volume forces, due to external fields, say gravity or electric fields for the case of charged fluids. The flux of momentum is a second order tensor, which makes the book-keeping more cumbersome, because there are several contributions to the momentum budget, namely: i) Inertial terms due to the motion of the fluid across the faces of cube, ii) pressure terms due to the component of the contact force normal to the surface it acts upon, iii) stress-terms, due to the tangential component of the force. To be noted that the latter two require independent inputs, namely an equation of state for the pressure as a function of density and temperature, as well as a constitutive equation for the stress σ_{ij} as a function of the strain $\nabla_i u_j$.

$$P = P(\rho, T)$$

$$\sigma_{ij} = \mu(\partial_i u_j + \partial_j u_i) + \lambda(\partial_k u_k)\delta_{ij}$$

where μ is the dynamic shear viscosity and λ associates with the bulk viscosity. Also to be noted that the latter does NOT conserve the total energy of the fluid (kinetic=mechanical+thermal plus potential). These are dissipative terms which emerge from Newtonian mechanics as a result of an approximation.

Lagrangian formulation

Go-with-the flow, Lagrangian form using the material derivative. The materials derivative runs along the material lines of the flow.

$$Du_i/ Dt \equiv \partial_t u_i + u_j \partial_j u_i$$

In a co-moving frame ($u_i = 0$) it reduces to the standard time derivative, which is a major simplification.

$$D\rho/ Dt = -\rho \partial_i u_i$$

$$Du_i /Dt = -(\partial_i P)/\rho + (\partial_j \sigma_{ij})/\rho$$

Each blob represents a multitude of molecules: a cc of air contains of the order of 1021 molecules (Loschmidt number). This formulation, which is the closest to Newtonian dynamics, is well suited to compressible flows with large deformations (say combustion enegines).

General features and family of flows

The NSE are: multi-dimensional, time-dependent, vectorial, non-linear and often live in complex geometries (car, airplanes, buildings ..). They are characterized by three main type of forces: Inertial, Pressure, Dissipation. The ratio between these three forces defines the main regimes of fluid flows, characterized by two main dimensionless parameters.

Mach (squared) = Inertia/Pressure;

Reynolds = Inertia/Dissipation

To be reminded that the ratio Ma/Re delivers the Knudsen number, which owes to be very small (less that some percent) if fluid dynamics is to apply at all. Low/High Mach characterize Incompressible/Compressible flows.Low/High Reynolds characterize Laminar/Turbulent flows Flows at zero Mach are called Stokes flows, a useful idealization for creeping flows in porous media and many biological applications. Flows with zero dissipation are called inviscid, a useful idealization for high Reynolds flows.

Compressible/Incompressible

Compressible fluids, typically gases, are characterized by a non-zero divergence of the velocity, i.e the volume occupied by a given amount of mass changes in space and time, so that density is alive. Recall that the divergence of the velocity field describes the change in volume of an element transported by the fluid. Positive(negative) divergence indicates expansion (compression). Zero divergence means that the volume is conserved (shape may change, even dramatically, though). Compressible flows support sound and shock waves. Sound waves usually carry small perturbations, shock waves carry the large ones, i.e. major density changes across very thin (molecular size) layers. Incompressiblity is tantamount to infinite sound speed: unphysical but neartrue for many fluids which undergo very small changes in density upon huge changes in pressure, parts per thousands or less. Water is a good example and so are most liquids.

$$Ma \to 0$$

Since the information travels at infinite speed, they encode action at distance, which is a tough constraint on numerical methods. Indeed incompressible flows require computational fluid techniques on their own.

Viscid/Inviscid

Inviscid fluids have strictly zero viscosity, v = 0 hence they do not dissipate (also called perfect fluids). The NSE with zero viscosity are called Euler equations. This is a (very useful) idealization, which describes accurately flows away from boundaries. Near boundaries, however, the approximation breaks down, because strong gradients couple even to vanishingly small viscosity and generate dissipation. Dissipation concentrates in thin regions called "boundary layers", often below mm for macroscopic flows in the scale of meters. However, neglecting these layers would be a deadly mistake because all energy is dissipated there. Thus the

limit $v \to 0$

must be kept very distinct from the strictly inviscid condition v = 0.

Laminar/Turbulent

The Reynolds number measures Inertia/Dissipation. It is typically a large number, easily in the order of millions for ordinary macroscopic bodies. Example: U = 30 m/s, L = 1m, v $=10^{-5}$ m2/s, gives Re ~ $30\times1\times10^5 = 310^6$ It is the ratio between the macroscopic size L and the molecular mean free path, as best appreciated via the Von Karman relation Re = M a Kn When the flow is very slow (creeping flows) the NSE are called Stokes equations:

i.e $Re \to 0$

Very relevant to biological flows, say cells in blood, bacteria in water... ZeroReynolds does not necessarily imply slow flows because the non-linearity can be suppressed (depleted) by mere geometrical symmetries, like in Coeutte or Poiseuille flows. The opposite limit to fully developed turbulence,

i.e. $Re \to \infty$.

Turbulent flows are ubiquituous and represent one of the most challenging problems of classical physics and engineering.

Newtonian/Non Newtonian

This refers to the relation between appled stress σij and resulting strain Dij ≡ ∂iuj . Linear proportionality characterizes Newtonian fluids.

Boundary Conditions

Boundary conditions select the solutions compatible with the environmental constraints. They depend both on the environment and the inherent nature/regime of the flow. In actual practice, they are the most critical factor in the development of robust and efficient CFD methods. Among others:

- Solid walls: no-slip velocity
- Inlets: imposed density (pressure) and velocity
- Outlets: imposed density(pressure) and zero normal gradient
- Symmetry Planes: zero normal gradient

It is easily recognized that a complete listing of fluid applications would be nearly impossible simply because the presence of fluids in technological devices is ubiquitous. The following provide some particularly interesting and important examples from an engineering standpoint.

1. Internal combustion engines—all types of transportation systems
2. Turbojet, scramjet, rocket engines—aerospace propulsion systems
3. Waste disposal (a) chemical treatment (b) incineration (c) sewage transport and treatment
4. Pollution dispersal—in the atmosphere (smog); in rivers and oceans
5. Steam, gas and wind turbines, and hydroelectric facilities for electric power generation
6. Pipelines (a) crude oil and natural gas transferral (b) irrigation facilities (c) office building and household plumbing
7. Fluid/structure interaction (a) design of tall buildings (b) continental shelf oil-drilling rigs (c) dams, bridges, etc. (d) aircraft and launch vehicle airframes and control systems
8. Heating, ventilating and air-conditioning (HVAC) systems
9. Cooling systems for high-density electronic devices—digital computers from PCs to supercomputers
10. Solar heat and geothermal heat utilization
11. Artificial hearts, kidney dialysis machines, insulin pumps
12. Manufacturing processes (a) spray painting automobiles, trucks, etc. (b) filling of containers, e.g., cans of soup, cartons of milk, plastic bottles of soda (c) operation of various hydraulic devices (d) chemical vapor deposition, drawing of synthetic fibers, wires, rods, etc.

We conclude from the various preceding examples that there is essentially no part of our daily lives that is not influenced by fluids. As a consequence, it is extremely important that engineers be capable of predicting fluid motion. In particular, the majority of engineers who are not fluid dynamicists still will need to interact, on a technical basis, with those who are quite frequently; and a basic competence in fluid dynamics will make such interactions more productive.

Theoretical studies of fluid dynamics generally require considerable simplifications of the equations of fluid motion mentioned above. We present these equations here as a prelude to topics we will consider in detail as the course proceeds. The version we give is somewhat simplified, but it is sufficient for our present purposes.

$\nabla \cdot U = 0$ (conservation of mass)

and

$DU/ Dt = -\nabla P + 1/ Re\nabla^2 U$ (balance of momentum). ($u,v,w)^T$

These are the Navier–Stokes (N.–S.) equations of incompressible fluid flow. In these equations all quantities are dimensionless, as we will discuss in detail later: $U \equiv (u,v,w)^T$ is the velocity vector; P is pressure divided by (assumed constant) density, and Re is a dimensionless parameter known as the Reynolds number. We will later see that this is one of the most important parameters in all of fluid dynamics; indeed, considerable qualitative information about a flow field can often be deduced simply by knowing its value.

In particular, one of the main efforts in theoretical analysis of fluid flow has always been to learn to predict changes in the qualitative nature of a flow as Re is increased. In general, this is a very difficult task far beyond the intended purpose of these lectures. But we mention it here to emphasize the importance of proficiency in applied mathematics in theoretical studies of fluid flow. From a physical point of view, with geometry of the flow situation fixed, a flow field generally becomes “more complicated” as Re increases. This is indicated by the accompanying time series of a velocity component for three different values of Re.

We also point out that the N.–S. equations are widely studied by mathematicians, and they are said to have been one of two main progenitors of 20th-Century mathematical analysis. (The other was the Schrodinger equation of quantum mechanics.) In the current era it is hoped that such mathematical analyses will shed some light on the problem of turbulent fluid flow, often termed “the last unsolved problem of classical mathematical physics.” We will from time to time discuss turbulence in these lectures because most fluid flows are turbulent, and some understanding of itis essential for engineering analyses. But we will not attempt a rigorous treatment of this topic. Furthermore, it would not be be possible to employ the level of mathematics used by research mathematicians in their studies of the N.–S. equations. This is generally too difficult, even for graduate students.

Experimental fluid dynamics

In a sense, experimental studies in fluid dynamics must be viewed as beginning when our earliest ancestors began learning to swim, to use logs for transportation on rivers and later to develop a myriad assortment of containers, vessels, pottery, etc., for storing liquids and later pouring and using them. Rather obviously, fluid experiments performed today in firstclass fluids laboratories are far more sophisticated. Nevertheless, until only very recently the outcome of most fluids experiments was mainly a qualitative (and not quantitative) understanding of fluid motion. An indication of this is provided by the adjacent pictures of wind tunnel experiments. In each of these we are able to discern quite detailed qualitative aspects of the flow over different prolate spheroids. Basic flow patterns are evident from colored streaks, even to the point of indications of flow “separation” and transition to turbulence. However, such diagnostics provide no information on actual flow velocity or pressure—the main quantities appearing in the theoretical equations, and needed for engineering analyses. There have long been methods for measuring pressure in a flow field, and these could be used simultaneously with the flow visualization of the above figures to gain some quantitative data. On the other hand, it has been possible to accurately measure flow velocity simultaneously over large areas of a flow field only recently. If point measurements are sufficient, then hot-wire anemometry (HWA) or laser-doppler velocimetry (LDV) can be used; but for field measurements it is necessary to employ some form of particle image velocimetry (PIV). The following figure shows an example of such a measurement for fluid between two co-axial cylinders with the inner one rotating. This corresponds to a two-dimensional slice through a long row of toroidally-shaped (donutlike) flow structures going into and coming out of the plane of the page, i.e., wrapping around the circumference of the inner cylinder. The arrows indicate flow direction in the plane; the red asterisks show the center of the “vortex,” and the white pluses are locations at which detailed time series of flow velocity also have been recorded. It is clear that this quantitative detail is far superior to the simple visualizations shown in the previous figures, and as a consequence PIV is rapidly becoming the preferred diagnostic in many flow situations.

Computational fluid dynamics

We have already noted that CFD is rapidly becoming the dominant flow analysis technique, especially in industrial environments. The reader need only enter CFD in the search tool of any web browser to discover its prevalence. CFD codes are available from many commercial vendors and as freeware from government laboratories, and many of these codes can be implemented on anything from a PC (often, even a laptop) to modern parallel supercomputers. In fact, it is notdifficult to find CFD codes that can be run over the internet from any typical browser. Here we display a few results produced by such codes to indicate the wide range of problems to which CFD has already been applied, and we will briefly describe some of the potential future areas for its use. The figure in the lower left-hand corner provides a direct comparison with experimental results shown in an earlier figure. The computed flow patterns are very similar to those of the experiment, but in contrast to the experimental data the calculation provides not only visualization of qualitative flow features but also detailed quantitative output for all velocity component values and pressure, typically at on the order of 10^5 to 10^6 locations in the flow field. The upper left-hand figure displays predictions of the instantaneous flow field in the left ventricle of the human heart. Use of CFD in biomedical and bioengineering areas is still in its infancy, but there is little doubt that it will ultimately dominate all other analysis techniques in these areas because of its generality and flexibility. The center figure depicts the pressure field over the entire surface of an airliner as obtained using CFD. It was the need to make such predictions for aircraft design that led to the development of CFD, initially in the U. S. aerospace industry and NASA laboratories, and CFD was the driving force behind the development of supercomputers. Calculations of the type shown here are routine today, but as recently as a decade ago they would have required months of CPU time. The upper right-hand figure shows the temperature field and a portion (close to the fan) of the velocity field in a (not-so-modern) PC. This is a very important application of CFD simply because of the large number of PCs produced and sold every year worldwide. The basic design tradeoff is the following. For a given PC model it is necessary to employ a fan that can produce sufficient air flow to cool the computer by forced convection, maintaining temperatures within the operating limits of the various electronic devices throughout the PC. But effectiveness of forced convection cooling is strongly influenced by details of shape and arrangement of circuit boards, disk drives, etc. Moreover, power input to the fan(s), number of fans and their locations all are important design parameters that influence, among other things, the unwanted noise produced by the PC. Finally, the lower right-hand figure shows pressure distribution and qualitative nature of the velocity field for flow over a race car, as computed using CFD. In recent years CFD has played an ever-increasing role in many areas of sports and athletics—from study and design of Olympic swimware to the design of a new type of golf ball providing significantly longer flight times, and thus driving distance (and currently banned by the PGA). The example of a race car also reflects current heavy use of CFD in numerous areas of automobile production ranging from the design of modern internal combustion engines exhibiting improved efficiency and reduced emissions to various aspects of the manufacturing process, per se, including, for example, spray painting of the completed vehicles. It is essential to recognize that a CFD computer code solves the Navier–Stokes equations, given earlier, and this is not a trivial undertaking—often even for seemingly easy physical problems. The user of such codes must understand the mathematics of these equations sufficiently well to be able to supply all required auxiliary data for any given problem, and he/she must have sufficient grasp of the basic physics of fluid flow to be able to assess the outcome of a calculation and determine, among other things, whether it is "physically reasonable"—and if not, decide what to do next.

We present the Navier-Stokes equations (NSE) of continuum fluid dynamics. The traditional approach is to derive teh NSE by applying Newton's law to a finite volume of fluid. This, together with condition of mass conservation, i.e. change of mass per unit time equal mass flux in minus mass flux out, delivers the NSE in conservative form, also known as Eulerian form, as it refers to the massmomentum balance as drawn by an observer at rest. The other approach, known 1 as Lagrangian, corresponds to the picture taken by a co-moving observer (gowith-the flow). Both approaches have merits and pitfalls, but the conservative form is generally more popular, especially for incompressible flows The Navier Stokes equations are non-linear vector equations, hence they can be written in many different equivalent ways, the simplest one being the cartesian notation. Other common forms are cylindrical (axial-symmetric flows) or spherical (radial flows). In non-cartesian coordinates the differential operators become more cumbersome due to metric terms (inertial forces). For geometries of real-life complexity (cars, airplanes, ...) no global coordinate system can be used, and one resorts to non-coordinate based representations, such as finite-volumes and finite-elements.

5. Conclusion

The Navier-Stokes equations of continuum fluid mechanics are simply Newton's law ma = F as applied to a small volume of fluid. Despite their elementary physical meaning, they prove exceedingly difficult to solve, as they assemble three nigthmares of computational physics: strong non-linearity, complex geometry, fully three-dimensional, time-dependent configurations. The theoretical aspects of fluid dynamics will be studied; it is simply not possible to cover all three facets of the subject of fluid dynamics in a single one-semester course meeting only three hours per week. At the same time, however, we will try at every opportunity to indicate how the theoretical topics we study impact both computational and experimental practice. Moreover, the approach to be taken in these lectures will be to emphasize the importance and utility of the "equations of fluid motion" (the Navier–Stokes equations). There has been a tendency in recent years to de-emphasize the use of these equations at precisely a time when they should, instead, be emphasized. They are the underlying core of every CFD analysis tool, and failure to understand their physical origins and how to deal with their mathematical formulations will lead to serious deficiencies in a modern industrial setting where CFD is heavily used. The equations of fluid motion (often simply termed "the governing equations") consist of a system of partial differential equations (as can be seen in an earlier section) which we will derive from basic physical and mathematical arguments very early in these lectures. Once we have these equations in hand, and understand the physics whence they came, we will be able to very efficiently attack a wide range of practical problems. But there is considerable physical and notational background needed before we can do this, and it is this material that will be emphasized in the second chapter of these lectures. There are four main physical ideas that form the basis of fluid dynamics. These are: i) the continuum hypothesis, ii) conservation of mass, iii) balance of momentum (Newton's second law of motion) and iv) balance (conservation) of energy. The last of these is not needed in the description of some types of flows, as we will later see, and it will receive a less detailed treatment than items ii) and iii) which are crucial to all of what we will study in this course. The remaining chapters of this set of lectures contain the following material. we discuss the first of the main physical ideas noted above, provide a precise definition of a fluid, and then describe basic fluid properties, classifications and ways in which fluid flow can be visualized, e.g., in laboratory experiments. This is devoted to a simple mathematical derivation of the Navier–Stokes equations starting from the fundamental physical law and followed with further discussion of the physics embodied in these equations, but now in the context of a precise mathematical representation. We then consider "scaling" the N.–S. equations, i.e., putting them in dimensionless form, and we study the corresponding dimensionless parameters. Then we turn to applications, both theoretical/analytical and practical. In particular, we will study some of the simpler exact solutions to the N.–S. equations because these lead to deeper physical insights into the behavior of fluid motions, and we will also expend significant effort on such topics as engineering calculations of flow in pipes and ducts because of the extensive practical importance of such analyses.

References

1. Batchelor, G.K. 2000. An Introduction to Fluid Dynamics. Cambridge.
2. Emanuel, G. 2000. Analytical Fluid Dynamics. CRC Press.
3. Faber, T.E. 1995. Fluid Dynamics for Physicists, 1st Edition. Cambridge.
4. Lamb, H. 1993 Hydrodynamics, 6th Edition. Cambridge.
5. Murray, C.D., and Dermott, S.F. 1999. Solar System Dynamics. Cambridge.

6. Schlichting, H. 1987. Boundary Layer Theory, 7th Edition. McGraw-Hill.
7. Batchelor, G. (1969). Physics of Fluids 12(12): II–223. Berloff, N. G. (2004). J. Phys. A 37: 1617.
8. Blaauwgeers, R., Eltsov, V. B., Eska, G., Finne, A. P., Haley, R. P., Krusius, M., Ruohio, J. J., Skrbek, L. & Volovik, G. E. (2002).
9. Batchelor, G.K. Introduction to Fluid Dynamics, Cambridge Univsrsity Press 1967
10. Prandtl, L. Essentials of Fluid Dynamics, Hafner 1952.
11. Courant and Freidrichs, Supersonic Flow and Shock Waves, Interscience 1948.
12. Meyer, An Introduction to Mathematical Fluids Dynamics, Dover 1971.
13. D. J. Acheson, Elementary Fluid Dynamics, Clarendon 1990.
14. Lamb, Hydrodynamics (6th Ed.), Cambridge University Press 1932

