cupress

Computational Fluid Dynamics

by

Finite Element and Finite Volume Methods

Pramote Dechaumphai

Computational Fluid Dynamics

by

Finite Element and Finite Volume Methods

Computational Fluid Dynamics

by

Finite Element and Finite Volume Methods

Pramote Dechaumphai

Chulalongkorn University press

2026

350.-

Dechaumphai, Pramote

Computational fluid dynamics by finite element and finite volume methods / Pramote Dechaumphai

1. Computational fluid dynamics. 2. Fluid dynamics. 3. Finite element method

4. Finite element method -- Data processing. 5. Finite volume method.

620.1064

ISBN (e-book) 978-974-03-4438-4

CUP. 2730

Knowledge to All
www.cupress.chula.ac.th

Published by **Chulalongkorn University Press**

First edition 2026

www.cupress.chula.ac.th [CUB6711-001]

Tel. 0-2218-3562-3

Contact: **Chulalongkorn University Book Center**

Phyathai Road, Pathumwan District, Bangkok 10330, Thailand

http://www.chulabook.com

Tel: 08-6323-3703-4

customer@cubook.chula.ac.th, info@cubook.chula.ac.th

Apps: CU-eBook Store

Preface

The textbook "Computational Fluid Dynamics by Finite Element and Finite Volume Methods" has been translated from the latest Thai edition (4th edition). It is published and distributed in both hardcopy and e-book formats by Chulalongkorn University Press in Bangkok, Thailand.

This textbook is divided into four main parts. The first part covers the fundamentals of fluid flows and the derivation of the Navier-Stokes equations. The second part presents the finite element formulations for four types of fluid flows: (a) inviscid incompressible flow, (b) inviscid compressible flow, (c) viscous incompressible flow, and (d) viscous compressible flow. The third part revisits these four types of fluid flows, but using the finite volume method. The final part covers the characteristics and significance of turbulent flow, explores various turbulence models, and discusses numerical simulation techniques. The book includes numerous basic examples that are easy to understand, along with computer programs in MATLAB, Mathematica, and Fortran. It also contains a large number of application problems.

The computer programs and files mentioned in this book can be downloaded from the following website: https://goo.gl/57hUHE.

The author extends heartfelt gratitude to his esteemed former Professor, Dr. Earl A. Thornton, and his supervisor, Dr. Allan R. Wieting, from the Aerothermal Loads Branch at NASA Langley Research Center. Appreciation is also extended to the students at NASA Langley Research Center, Old Dominion University, and Chulalongkorn University who actively participated in the finite element and finite volume method courses he offered.

Special thanks go to Chulalongkorn University Press for their role in publishing the book, contributing to its dissemination and impact. Lastly, the author expresses deep appreciation to his wife, Mrs. Yupa Dechaumphai, for her understanding and unwavering support throughout the writing process, acknowledging the significant role she played in bringing the book to fruition.

Pramote Dechaumphai

Contents

Part I

Numerical Methods and Fluid Flow Fundamentals

Chapter 1

Computational Fluid Dynamics

1.1 Introduction

The use of Computational Fluid Dynamics (CFD) has become increasingly crucial in both engineering design and scientific research for analyzing flow behaviors such as velocity, pressure, and temperature. CFD integrates the application of numerical methods and computer technology to solve partial differential equations that describe the dynamics of fluid flow. This approach yields results that can be visually represented through color graphics, greatly enhancing the ability of analysts to comprehend complex flow phenomena. Consequently, CFD facilitates the refinement and optimization of design models, allowing for the validation of designs on computer screens before proceeding to actual construction or further experimental validation. This preventive verification process boosts confidence in the design's feasibility and performance.

Incorporating CFD into the design process markedly diminishes both costs and timeframes when compared to traditional experimental methods that dominated earlier engineering practices. A prime example of this shift can be seen in the development of early Boeing aircraft models intended for commercial transcontinental flights. Previously, engineers were tasked with constructing scale models of the airplanes from wood and testing these models in wind tunnels to analyze airflow conditions, including lift generated by the wings and the plane's underbelly. This method was not only labor-intensive but also required significant investment in materials and experimental setups, extending the development cycle from construction to testing over years. Furthermore, should the testing reveal any design flaws, such as wings that were too large or insufficiently long, the process of making corrections was cumbersome. Adjustments necessitated additional time for modifications and subsequent testing, which, although occasionally leading to improvements, did not always result in the sought-after enhancements.

By leveraging CFD, the design timeline and experimental costs associated with aircraft development can be drastically reduced. Engineers proficient in CFD can effortlessly generate simulation models of Boeing aircraft to analyze airflow conditions, a process depicted in Figure 1.1. A significant benefit of this approach is the agility with which modifications can be applied to the aircraft's design, such as alterations to the wing configuration or other components. Additionally, CFD enables rapid acquisition of detailed flow field information in specific areas, a task that could be exceedingly challenging or time-intensive with traditional experimental methods.

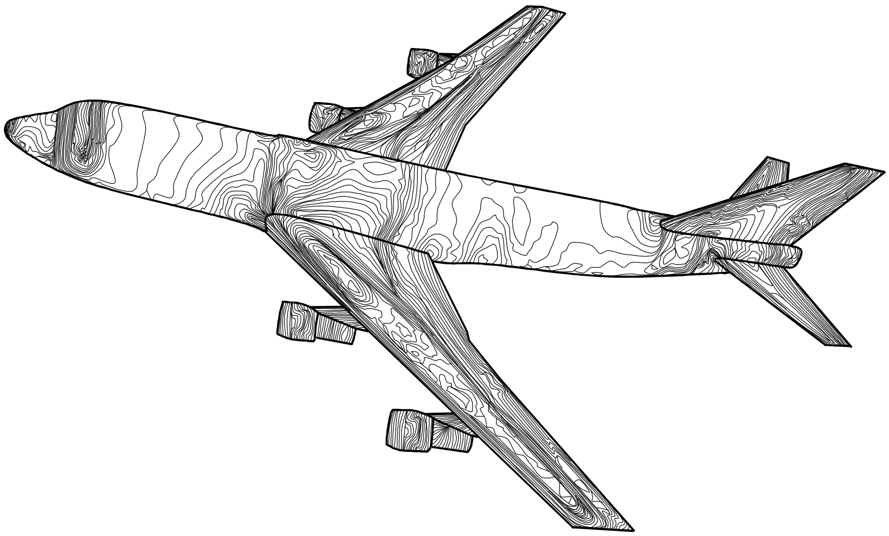


Figure 1.1 Contour lines illustrate pressure distribution on aircraft surfaces.

Moreover, the versatility of CFD extends beyond aircraft models. The same software used for assessing flow conditions around the Boeing aircraft can also be applied to analyze airflow through other intricate shapes, such as determining air conditions around fighter jets, illustrated in Figure 1.2. These examples underscore the value of CFD in providing detailed insights into pressure variations across different sections of the aircraft, from the nose, along the wings, to the vicinity of underwing areas. This capability not only enhances design precision but also streamlines the development process, illustrating the comprehensive advantages of CFD in modern aerospace engineering.

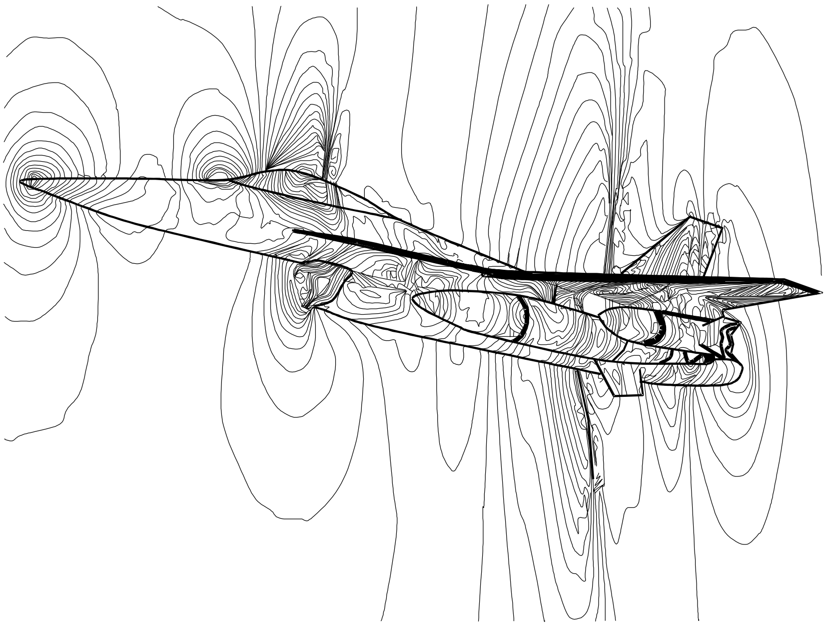


Figure 1.2 Contour lines illustrate pressure distribution on the surface and around a fighter jet aircraft.

Addressing modern challenges in design often confronts the limitation that many contemporary issues are impractical to test experimentally, or such testing would entail prohibitively high expenses. Take, for instance, the development of an aerospace plane designed for high-speed travel, capable of circumnavigating half the globe in merely two hours at velocities reaching 25 times the speed of sound, as illustrated in Figure 1.3. At these extraordinary speeds, the friction with the atmosphere generates heat up to 70,000 Btu/ft^2-sec. This is starkly contrasted with the mere 0.12 Btu/ft^2-sec one might experience from sunlight on a particularly bright day. The significant heat produced through atmospheric friction has the potential to melt the spacecraft's external surface.

The most extensive continuous experiments on the ground are capped at 2 minutes duration, achieving speeds only up to 8 times the speed of sound within a wind tunnel at NASA Langley Research Center, the place where the author previously worked. Conducting each experiment demands an enormous energy input to drive hot gas at high velocities across a spacecraft component under test. This process induces severe vibrations in the facility, necessitating the suspension of all other activities during the 2-minute test interval. Furthermore, the average cost for equipment and labor per test approaches several thousand US dollars, underscoring the significant resource investment required for such pioneering explorations in aerospace design.

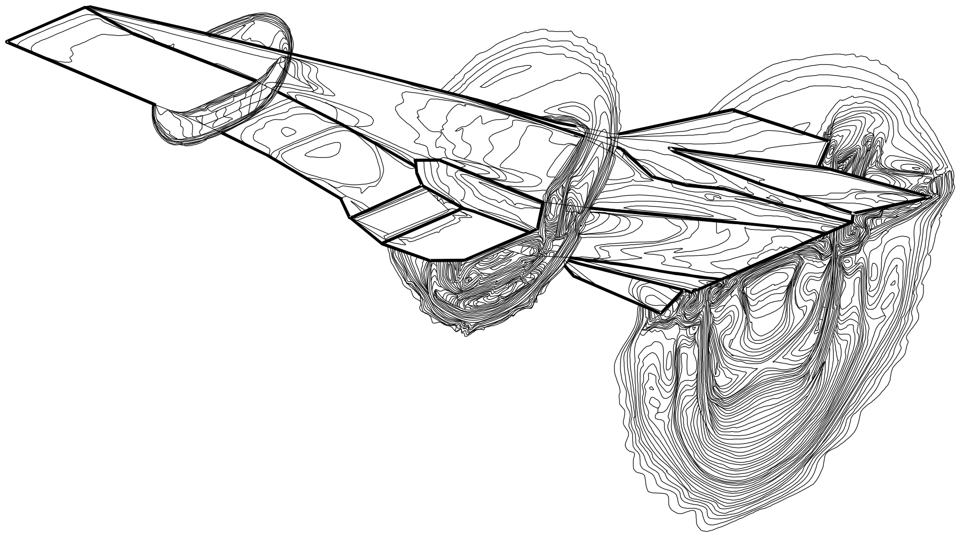


Figure 1.3 Contour lines illustrate pressure distribution on the surface and around the National Aerospace Plane.

Beyond addressing the overt challenges outlined previously, CFD has been instrumental in tackling various issues that enhance our current quality of life, such as analyzing ventilation efficiency within energy-saving buildings. Through experiments conducted in different scenarios of window operation for ventilation purposes, insightful data have been gathered. Figure 1.4 illustrates the airflow derived from these calculations, demonstrating how air impacts the building's left side. Some of this air traverses over the roof, while another portion enters through an open balcony door on the second floor's left side, circulates through the building, and exits upwards through an open window on the right side of the roof. Figure 1.5 depicts a similar setup for doors and windows as seen in the first instance, but with an additional door opened on the right side of the lower floor to enhance ventilation. The airflow conditions showcased in Figures 1.4 and 1.5, obtained through CFD, demonstrate the capability of this technology to efficiently model and predict the effects of various window configurations on air circulation in a relatively short time frame.

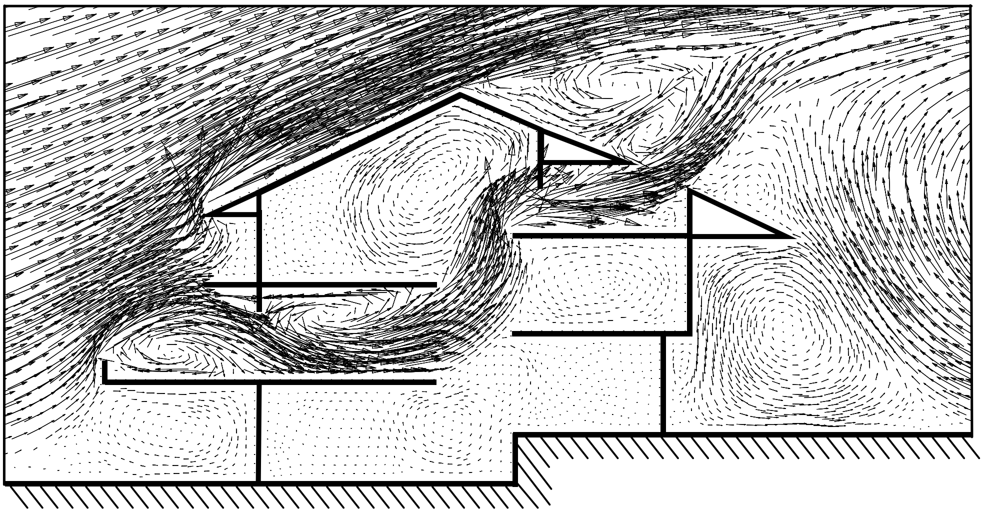


Figure 1.4 Velocity vectors depicting airflow through a house with an open skylight.

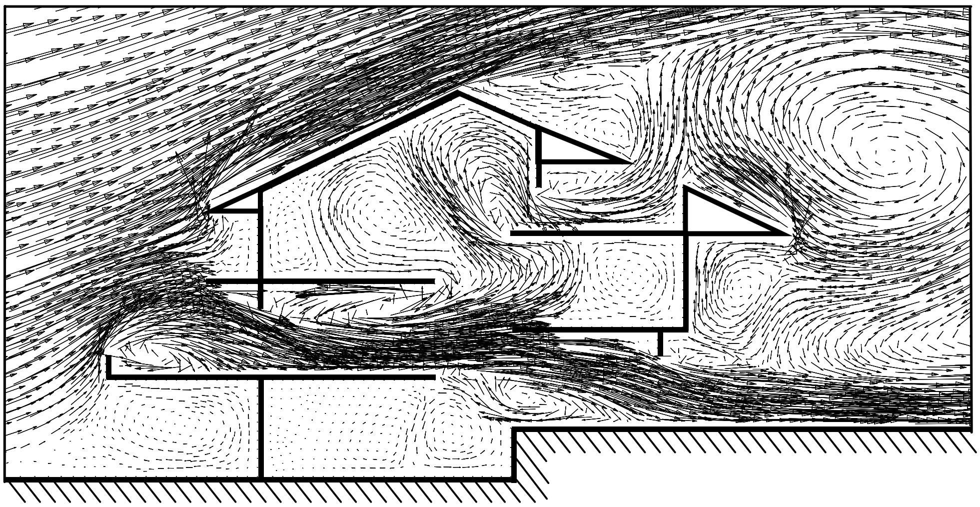


Figure 1.5 Velocity vectors depicting airflow through a house with an open skylight and a porch door.

CFD has become a cornerstone in engineering, extensively applied across a broad spectrum of research and development initiatives to amplify operational efficiency. This includes optimizing impeller designs in pumps for enhanced thrust, refining fuel distribution for uniform and complete combustion in engine cylinders, and shaping ship hulls to minimize water resistance during navigation. Furthermore, CFD aids in assessing the spread of wastewater in rivers, ensuring effective ventilation in electronic devices through microchip cooling to avert overheating, and providing precise weather forecasts for vast regions or specific microclimates within air-conditioned office spaces. An example of CFD's application is depicted in Figure 1.6, which visualizes the airflow in a 5×4×3 meter room, featuring a 1.5-meter tall cabinet centrally located. The walls of this room experience varied heat loads, and an air conditioner, positioned at one ceiling corner, emits air at 10°C, illustrating the detailed insight CFD offers into environmental and mechanical engineering challenges.

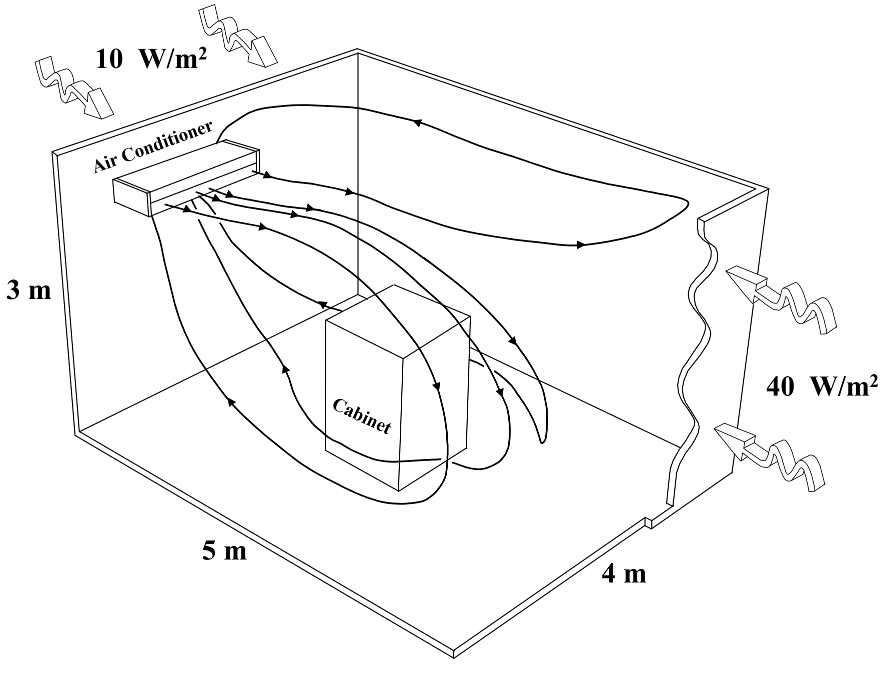


Figure 1.6 Particle tracing paths in an air-conditioned room with a cabinet placed in the middle of the room.

Figure 1.7 illustrates the temperature distribution across the same room's cross-section, demonstrating the application of CFD to effortlessly predict the flow conditions within. Moreover, CFD extends its utility beyond traditional engineering, facilitating research in scientific fields where physical experiments are either not feasible or potentially dangerous. This includes, but is not limited to, simulating the blood flow in veins and the heart, as well as modeling the interaction of hazardous chemicals.

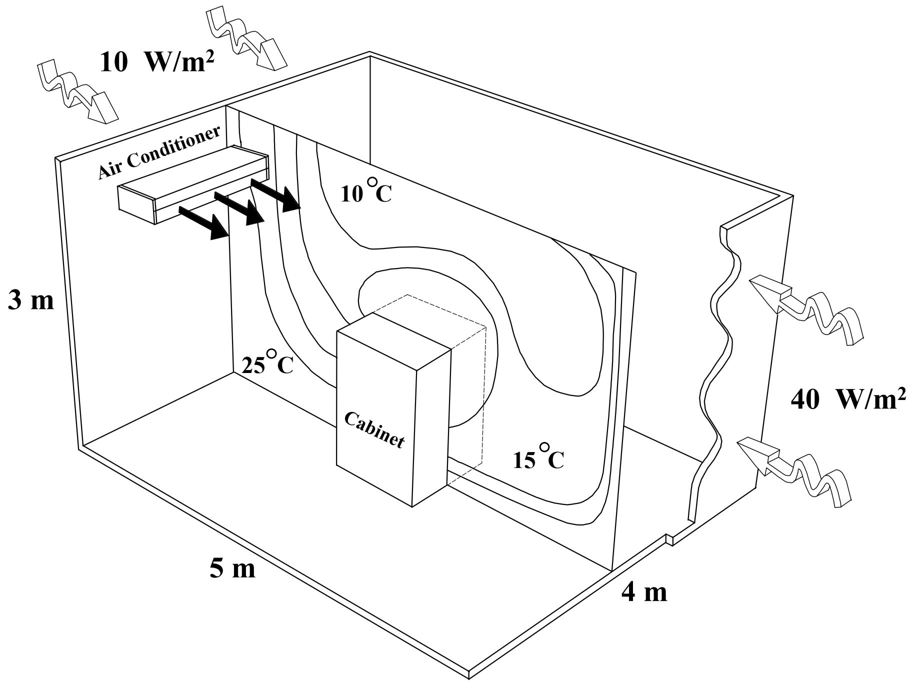


Figure 1.7 Temperature distribution in an air-conditioned room with a cabinet placed in the middle of the room.

The examples highlighted above demonstrate the profound impact of CFD across a wide spectrum of flow-related challenges. By significantly cutting down on the time and expenses associated with design processes, CFD enhances analytical capabilities and streamlines the experimental phase, particularly when physical experimentation is impractical or impossible. Furthermore, CFD facilitates a granular examination of flow dynamics in targeted areas, enabling the acquisition of accurate results and insights. Crucially, these instances underscore CFD's capacity to unlock extensive possibilities for tackling diverse flow issues, spanning practical applications to research and development initiatives.

1.2 Process of Solving Problems

Analyzing engineering problems, ranging from solid mechanics like stress analysis in structures and components, to heat transfer phenomena such as temperature distribution in cooling systems of transformers and motors, to fluid dynamics like the study of airflow in air-conditioned environments, or even tackling problems in diverse scientific areas, depends on three major factors or components. These are: (a) the partial differential equations that describe the reality of the problem, (b) the boundary conditions for the problem under study, and (c) the geometry or shape of the problem. If any of these components changes, the resulting outcomes will also change accordingly. A deep understanding of these three components is crucial for problem analysis through computation. The importance of each component that needs to be understood can be summarized as follows.

1.2.1 Governing Differential Equations

Most of engineering problems are governed by partial differential equations that vary in complexity. Partial differential equations are considered the core that represents the reality of each problem. For example, analyzing solid mechanics problems requires starting from a set of partial differential equations that describe the equilibrium of forces in three directions, which are always valid at any point in the problem. Heat transfer analysis must begin with partial differential equation that describe the equilibrium of heat transfer, while fluid dynamics analysis must start from a set of partial differential equations that represents the conservation of mass, momentum, and energy. These partial differential equations consist of various terms in derivative form, using symbols similar to an inverted of the number "six". As an example, the set of partial differential equations for two-dimensional steady-state viscous incompressible flow without considering temperature change (the details and meanings of each term will be presented in the later chapter) are,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1.1}$$

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) + \frac{\partial p}{\partial x} = \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{1.2}$$

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) + \frac{\partial p}{\partial y} = \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{1.3}$$

The three differential equations above consist of three unknown dependent variables that vary with the independent variables x and y. These three dependent variables are the velocity u in the x-direction, the velocity v in the y-direction, and the pressure p, all of which are functions of both x and y-coordinates. The symbols ρ and μ represent the density and the viscosity of the fluid, respectively.

Solving the above set of differential equations for a complex flow problem in order to find an exact solution is generally impossible. This is due to two main reasons. The first reason is that these partial differential equations are coupled, meaning that the solutions of the velocity components u, v and the pressure p must satisfy all equations. The second difficulty is that the partial differential equations

(1.2) and (1.3) are nonlinear since the unknown variables u and v appear as coefficients in the first terms of both equations. Such the set of nonlinear differential equations does not only makes finding an exact solution impossible but also greatly complicates the use of numerical methods for solving them.

These two main reasons significantly slow down the development of solutions for fluid flow problems compared to those for solid mechanics or heat transfer problems. The non-linear nature of the partial differential equations in fluid flow problems can lead to extremely complex outcomes when compared to solid or heat transfer problems, which are often linear and result in simpler, more straightforward distributions of solutions. For example, applying a certain force on a beam results in a stress on that beam. If the force applied to the beam is increased, the stress in the beam also increases in a proportional manner. However, in the case of fluid flow problems governed by the non-linear partial differential equations, it is different. As an example, turning on an air conditioner at a low speed in a room, it creates a certain circulation pattern of air. But if the air speed of the air conditioner is increased, the circulation pattern of the air in the room might change completely. That is the circulation does not just remain in the same pattern with an increased speed. Understanding the implications of the partial differential equations is thus critically important. Those looking to solve such problems must first understand and familiarize themselves with these equations, not just viewing them as mere mathematical equations, but also understanding their physical significance. This positive attitude towards solving these partial differential equations correctly can lead to meaningful and insightful outcomes that enhance the understanding of the problems at hand.

1.2.2 Boundary Conditions

In the process of solving the set of partial differential equations, boundary conditions are a crucial component that leads to appropriate solutions. If the problem being analyzed is unsteady, initial conditions must also be applied. Changing boundary conditions lead to changing solutions as well. Figure 1.8 illustrates the temperature distribution at the mouth of the Chao Phraya River during the flood season. The boundary condition for this problem is set so that water flows from the upper left corner at a speed of 3 meters per second and at a temperature of 25°C. On the upper right side, hot water at a temperature of 40°C is released from a factory, causing downstream temperatures to rise, which may affect the damage to aquatic animal farming in the downstream area. Figure 1.9 (a) shows the details of the temperature level contours in a small frame of Figure 1.8, where hot water from the factory is released and mixed with the cold river water near the Butterfly Island. Due to the flood season, the hot water from the factory is quite quickly dispersed. Figure 1.9 (b) also shows the solutions of the temperature distribution from the mixing of water near the Butterfly Island but in the dry season case. That is, it is assumed that the cold water in the Chao Phraya River flows from the upper left corner of the image at a reduced speed of 1.5 meters per second, resulting in a generally higher temperature in a broader area.

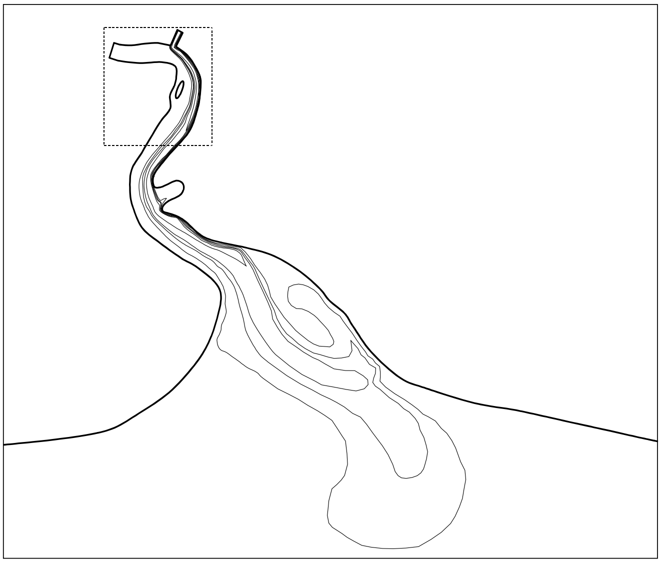
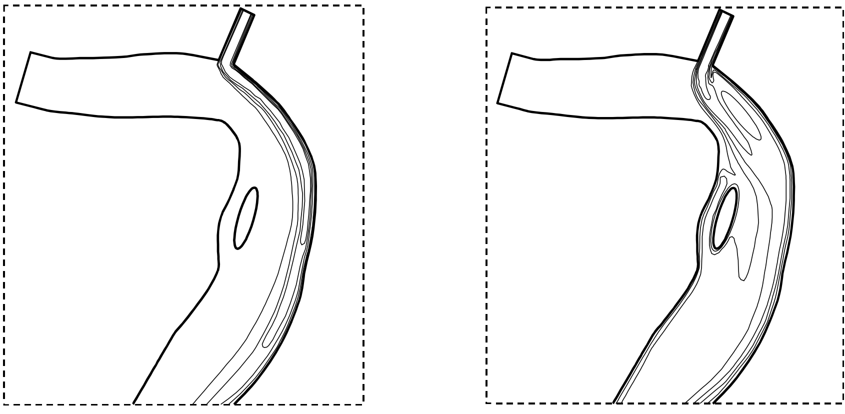


Figure 1.8 Contour lines indicate the temperature distribution at the mouth of the Chao Phraya River due to the factory releasing hot water during the flood season.



(a) Flood season

(b) Dry season

Figure 1.9 Details of contour lines show the temperature distribution from the mixing of waters at different temperatures in the area around the Butterfly Island.

The different temperature distributions shown in Figure 1.9 result from setting different boundary conditions. The specification of boundary conditions for flow problems in computational fluid dynamics depends on the type of partial differential equations, whether they are elliptic, parabolic, or hyperbolic, which in turn depends on the flow velocity. For example, flow at speeds greater than the speed of sound around a blunt body creates a bow shock wave positioned in front of the blunt body. In the area between the bow shock wave and the blunt body, the flow speed is lower than the speed of sound, making the differential equation elliptic in nature. However, just beyond this region, where the flow speed increases to exceed the speed of sound, the differential equation becomes hyperbolic. Such problems have historically been challenging to solve to obtain results, as the partial differential equation system can change from one type to another within the same computational domain. Therefore, in the chapters on different types of flows that will be presented, the boundary conditions will be clearly explained to facilitate understanding and to correctly and appropriately set the boundary conditions for each type of flow.

1.2.3 Geometries

In general basic problem-solving in classrooms, the geometry or shape of the problem is often not considered. Most studies focus only on the steps for solving the differential equations along with given boundary conditions. If the shapes of the problem are specified, they are typically just rectangles or squares. Not considering the problem shapes leads students to almost completely overlook the complexities of solving problems arising from the geometry. In engineering and science, problems are always designed with complex geometries, such as airflow through a house with rooms at different levels under a sloping roof, as shown in Figures 1.4 and 1.5. Problems may inherently have complex geometries, like the flow in the area near the mouth of the Chao Phraya River in Figures 1.8-1.9. The geometry, which covers changing flow areas, results in changing the flow solutions, even though the set of partial differential equations and boundary conditions remain the same. Figure 1.10 shows a flow through a narrow channel with a linear expansion, causing a small area of circulation in the lower left. Figure 1.11 shows the results from solving the same set of partial differential equations and boundary conditions near the entrance similar to Figure 1.10, but the geometry in the lower left of the problem has been expanded further, resulting in an increased area of circulation.

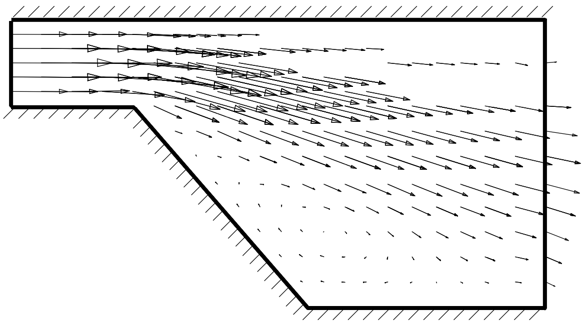


Figure 1.10 Velocity vectors indicate the flow behavior through a narrow channel geometry with a linear expansion.

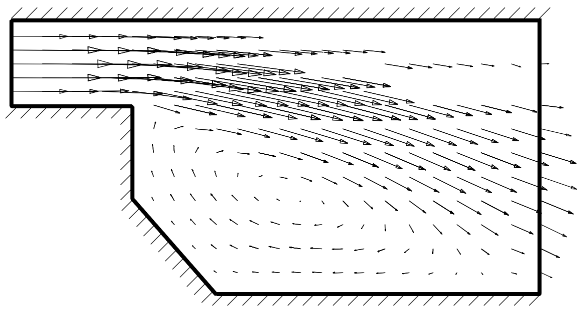


Figure 1.11 Velocity vectors indicate the flow behavior through a narrow channel geometry with a complicated expansion.

From these examples, it can be seen that the flow solutions are diverse, depending on the three main components mentioned above. These examples also demonstrate that if an efficient method for obtaining the solutions of these flow problems can be found, the analyst can modify the boundary conditions or the shape of the problem, which will lead to the desired outcomes. This idea makes CFD highly beneficial for design work nowadays.

The knowledge of problem-solving methods combined with the potential of solving problems using computers has led to the development of numerous commercial software. The primary function of these software is to find solutions by solving the three main components as described above. These software are generally expensive, and users must have sufficient knowledge in mathematics and computational methods to ensure that the solutions obtained from these software are accurate. It must be emphasized that these software are not like graphic software used for drawing or drafting. Using a Computer Aided Design (CAD) software to draw a circle on a computer screen and seeing it as the desired circle allows users to know they have the correct result. However, using a CFD software that provides the flow solutions, which are displayed in forms of velocity vectors and pressure in colors, requires deep understanding. If users do not understand what the software is doing internally, they cannot be sure that the solutions are accurate.

The operational steps of any CFD software are to solve a set of partial differential equations of flow, with the appropriate boundary conditions and the given problem geometry. Generally, every software follows the three main steps: (a) the step of defining the problem geometry and boundary conditions, which is called the pre-processing; (b) the analysis, which is the heart of the software; and (c) the post-processing for displaying the solutions obtained from the calculations.

(a) <u>Pre-processing</u> The initial process begins with the creation of the flow domain to be analyzed, consisting of steps ranging from creating lines, creating surfaces, to creating volumes if it's a three-dimensional flow. Then, this created flow domain is divided into small elements, where these elements are connected at nodes. The nodes are the positions to calculate the flow solution values such as the velocity components, pressure, and temperature. Finally, the properties of the fluid and the boundary conditions for the problem are defined accordingly.

This initial process, although it can be easily performed in commercial software today, often takes a considerable amount of time due to the complexity of the geometry of flow problems. For example, the flow over an engine under the wing attached to the aircraft's fuselage, as shown in Figure 1.12, requires the flow domain to be created accurately. The surface of this flow domain must be smooth along the curved surfaces of the engine under the wing, the curved surface of the aircraft's fuselage, and also where these various curved surfaces intersect in three dimensions. Then, the created flow domain is divided into a large number of small elements to be used in the next analysis step.

(b) <u>Analysis</u> The analysis step is considered the heart of these commercial software. Dividing the flow domain into too many small elements results in an increased number of the nodes with more unknowns, which in turn increases the computational time and the amount of computer memory required. However, conversely, this leads to higher solution accuracy. Experienced analysts must therefore weigh the pros and cons before performing the calculation.

The concept of discretization, dividing the flow domain into small elements, is to enable the use of simple functions to represent the characteristics of the flow through these small elements. These functions are then substituted into the set of the governing partial differential equations, combined with certain mathematical processes to minimize errors, resulting in a large system of algebraic equations that must be solved by numerical methods.

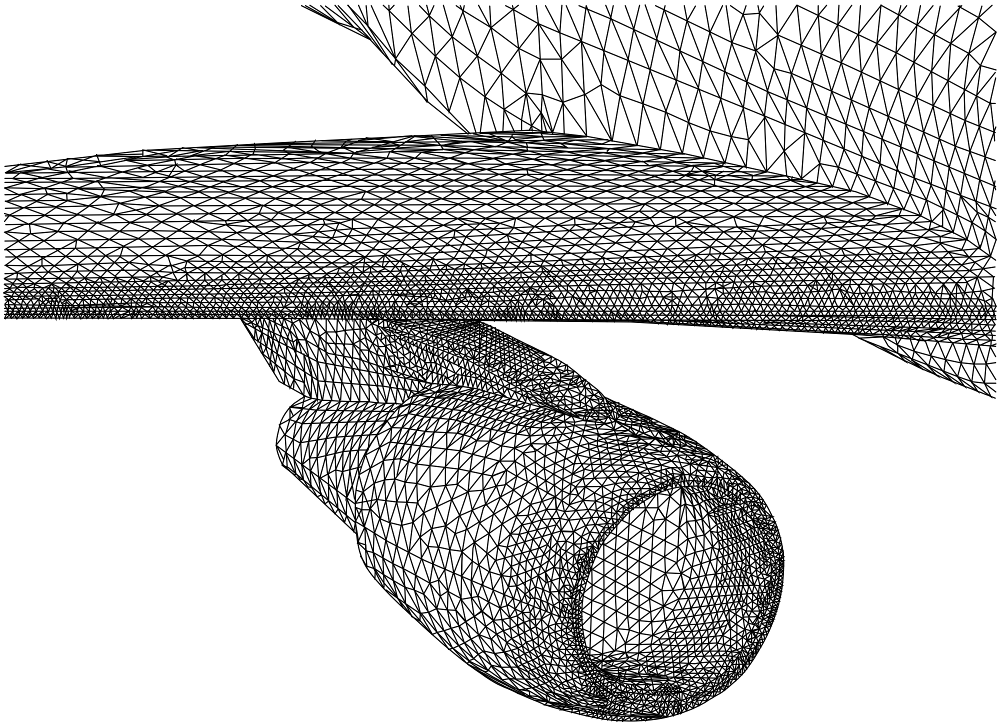


Figure 1.12 A finite element mesh on the surfaces of the engine under the wing attached to the aircraft fuselage.

The popular analysis methods include the Finite Difference Method (FDM), the Finite Element Method (FEM), and the Finite Volume Method (FVM). Currently, these three methods are integrated into the curriculum to assist in solving problems in various subjects at both the undergraduate and graduate levels for different fields in engineering and science.

In overview, the Finite Difference Method uses the Taylor series to approximate the derivative terms appearing in partial differential equations with values at the junction point under consideration and at adjacent junction points. This results in a system of algebraic equations that must be solved using the aforementioned numerical methods. The Finite Difference Method comprises straightforward steps that are easy to understand, but it has the limitation of not being easily applicable to problems with complex geometries.

At the same time, if the Finite Element Method or the Finite Volume Method is used, problems of any complex two-dimensional shape can easily be divided into elements such as triangles, irregular quadrilaterals. These elements do not need to be arranged in an orderly fashion like the rectangular grid required in the Finite Difference Method. The feasibility of the Finite Element and Finite

Volume Methods has made them increasingly popular for solving various engineering problems, which often have complex geometries. However, both of these methods, such as the Finite Element Method, involve more complex steps of formulating internal equations. Starting with the creation of element interpolation functions and applying the method of weighted residuals, thereby generating finite element equations for each element before assembling these equations into a large system of equations that also must be solved by numerical methods.

(c) Post-processing Results from the flow analysis process typically consist of velocity components in different directions, pressure, and temperature at nodes in the flow domain. These nodal solutions are all in the form of numerical values. Visualizing these solutions in the form of graphics is thus essential, such as vector plots at every nodes throughout the flow domain as shown in figures 1.4 and 1.5 of air flowing through a house. Contour lines, for example, the temperature distribution in an air-conditioned room as shown in figures 1.7, and in the area of the Chao Phraya River mouth as shown in figures 1.8 and 1.9. These plots are displayed in different colors on a computer screen to help understand the problem more quickly and effectively. Additionally, the results can be presented in the form of particle tracing paths, as shown in figure 1.6, among others. These various formats can be viewed from multiple directions, including showing fluid motion to create a more realistic feeling.

1.3 Essential Knowledge

Capabilities of computer software for analyzing fluid dynamics problems, as described earlier, often leads analysts to believe that these software can easily obtain flow solutions for any problem. However, in reality, since these software are based on advanced mathematics and computational methods, those who can use these software correctly and efficiently must have sufficient knowledge. The necessary knowledge components can be categorized into five aspects: (a) understanding of the set of partial differential equations and the physical meaning of different terms in these equations, (b) knowledge of numerical methods, (c) knowledge of finite element or finite volume methods, (d) basic knowledge of computer programming, and (e) sufficient experience in using the software. These five knowledge components are like five doors that must all be opened; if any door is closed, it will limit the potential of the users to use these software, leading to issues such as longer analysis times or excessive memory usage than necessary, incorrect use of the software, unawareness of what the software is doing while it is executing, results with low accuracy. Worst of all, getting incorrect results or being unable to explain why the results are valid. The above five knowledge components will help users understand the problem-solving process and be confident in the solutions obtained. These five foundational knowledge aspects can be summarized as follows.

(a) Knowledge in the differential equations CFD software is fundamentally based on solving a set of partial differential equations. Therefore, it is necessary to study the specific set of partial differential equations that the software is designed to solve, and to understand the physical phenomena governing the flow problems. Moreover, understanding the physical significance of each term in the set of equations is crucial in indicating the capabilities and limitations of the software itself. The terms in these equations further indicate the depth of computational methods required and how this affects the time taken to solve specific problems. Thus, understanding the set of partial differential equations is essential before starting to use any CFD software.

(b) Good background on numerical methods Numerical methods have become a mandatory subject in the study of many engineering disciplines today. They are a fundamental knowledge and a necessary component for computation. The basics of numerical methods include knowledge of interpolation functions, numerical differentiation and integration, solving ordinary differential equations, and partial differential equations. Most importantly, it involves knowledge of various techniques used to solve systems of algebraic equations. As an example, to solve a set of n linear algebraic equations in the form,

$$\underset{(n\times n)}{[A]}\underset{(n\times 1)}{\{X\}} = \underset{(n\times 1)}{\{B\}} \tag{1.4}$$

where $\{X\}$ is a vector or column matrix composed of unknown values x_1, x_2, x_3, ….., x_n; $\{B\}$ is a vector containing n known values. In this case, $[A]$ is a square matrix consisting of $n\times n$ coefficients. In practical problem-solving, the time spent solving this system of equations significantly exceeds the time spent on other calculations. As an example, consider a three-dimensional flow problem that might consist of only 50,000 equations (10,000 nodes, with each node having 5 unknowns of velocities in three directions, pressure, and temperature), which is considered a relatively small problem today. The matrix $[A]$ would consist of a total of 2.5 billion coefficients. Assuming writing down such 50,000 equations on paper by hand, taking 1 second per coefficient, it would take approximately 80 years to write all of these down. This estimate does not even begin to address solving this set of equations.

The example mentioned illustrates the benefits of numerical methods, which can be efficiently used on current computers. Moreover, due to the set of partial differential equations, as shown in equations (1.1)-(1.3), which consist of nonlinear terms, the resulting system of the algebraic equations is also in a nonlinear form,

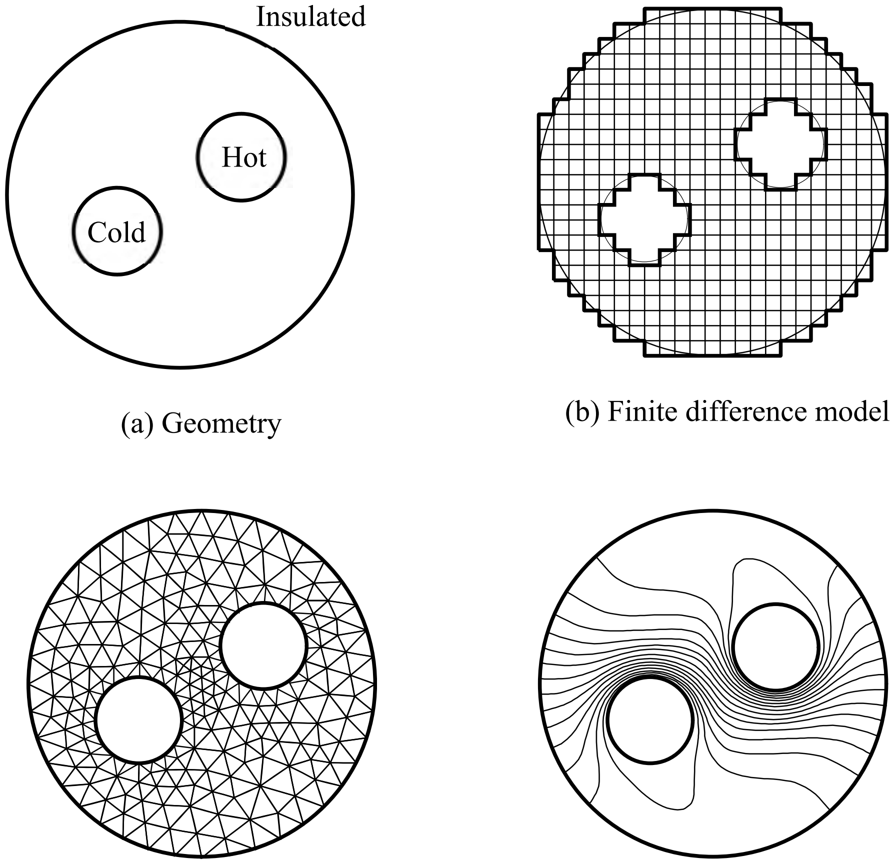
$$\underset{(n\times n)}{[A(x)]}\underset{(n\times 1)}{\{X\}} = \underset{(n\times 1)}{\{B\}} \tag{1.5}$$

The system of nonlinear algebraic equations above needs an iterative process, such as the Newton-Raphson iteration method, to solve for solutions. The foundation of this nonlinear equation solving technique is also included in the study of numerical methods.

(c) Knowledge in finite element and finite volume methods Analyzing any problem as described in section 1.2, the solution depends on three main components: the set of partial differential equations covering the problem, the boundary conditions, and the problem's geometry. The finite element and finite volume methods are techniques that fulfill these three main requirements. Starting with representing the problem's geometry using small elements, making the computation domain more accurate than the rectangular grid used with the finite difference method. Figure 1.13 (a) shows the computation domain on a cross-section of a large cylindrical pipe wrapped in insulation, with two smaller cylindrical pipes inside at high and low temperatures, filled with fluid between these pipe surfaces. The aim is to study the flow behavior and temperature of this fluid. Figure 1.13 (b) displays the rectangular grid using the finite difference method. The figure shows that the computation domain's boundaries cannot be accurately represented, appearing as steps along the pipe's curved surface. To reduce the size of these steps, it's necessary to densify the rectangular grid, resulting in an increased number of grid points and unknowns. Meanwhile, Figure 1.13 (c) illustrates the triangular elements created for the finite element method, which can better represent the pipe's curves, leading to more accurate results. Figure 1.13 (d) displays the contour lines of the temperature distribution obtained by the finite element method. The details of these results will be presented in Chapter 8.

After discretizing the computation domain into small elements, the finite element equations are then generated for each element. These equations are derived from the governing partial differential equations of the problem, combined with the use of the interpolation functions that approximate the unknown distributions over that element. This process will be described in detail in Chapter 3. Then, the equations of each element are assembled to form a large system of equations, which, in physical terms, is akin to assembling all elements together to form the actual shape or geometry of the problem. Subsequently, the boundary conditions of the problem are applied to this large system of equations, which is in the form of equations (1.4) or (1.5), before solving the entire system of equations by the numerical methods.

From the above explanation, it is evident that the finite element method fully meets the requirements of all three main components: the governing partial differential equations, the boundary conditions, and the geometry of the problem. Furthermore, since the finite element method is a numerical technique for finding an approximate solution by using simple functions to represent the unknown distribution, it implies that employing a more number of elements leads to higher solution accuracy.

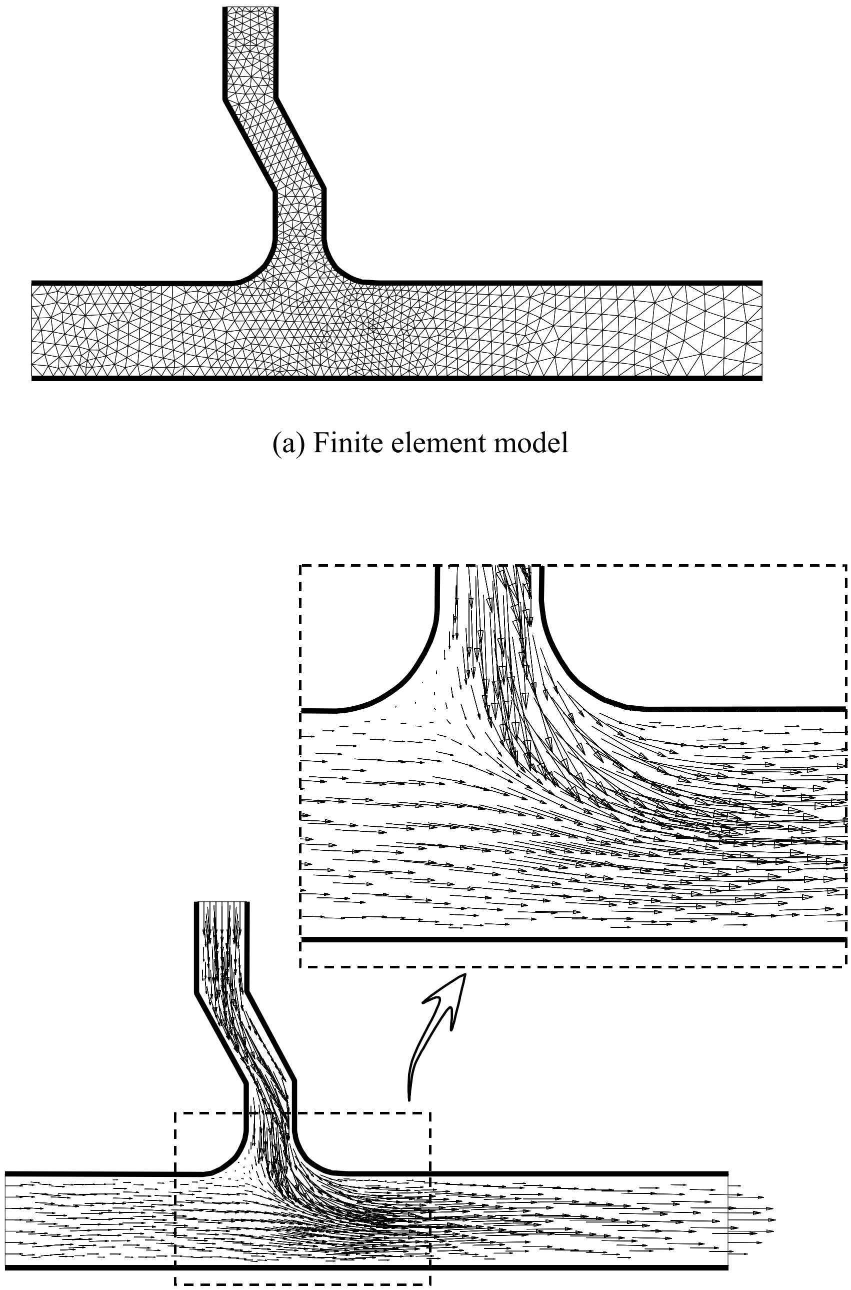


(a) Geometry (b) Finite difference model

(c) Finite element model (d) Temperature contours

Figure 1.13 Fluid temperature on pipe section with hot and cold tubes.

Figure 1.14 shows the use of the finite element method to solve the problem of flow in converging pipes. The problem includes a large horizontal pipe with flow entering from the left side. In the middle section of this large pipe, there is a smaller, curved pipe that converges and has flow entering from the top. Figure 1.14 (a) displays the finite element model, consisting of triangular elements that effectively represent the curved shape. Due to the complexity of the flow occurring where the two pipes converge, the elements in this area are made smaller to yield more accurate solutions in this region. Figure 1.14 (b) shows the flow behavior by the velocity vectors, with the details of the flow at the convergence of the pipes displayed in the top right insert.



(a) Finite element model

(b) Flow behavior by velocity vectors

Figure 1.13 Merging pipe flow by finite element method.

(d) Knowledge of computer programming Understanding the steps of computation, which are developed into the corresponding computer programs, is very important for analyzing problems. A significant number of users try to use off-the-shelf computer programs to solve complex flow problems. The computations may take several days to complete. Understanding the computational steps in these computer programs can lead to approaches that significantly reduce computational time while still achieving equally accurate solutions. A lack of understanding of the computational processes used in computer programs can sometimes lead to not achieving any solution at all after spending a large computational time, or, if results are obtained, they may not be accurate.

The reasons mentioned above indirectly compel program users to have a relatively profound understanding of the problem analysis process. This understanding can be acquired from classes or from having solved similar problems several times before. Therefore, when purchasing these computer programs or software from vendors, the vendors often include training classes to at least provide a basic understanding of how to correctly use the software. However, the best method remains to accumulate fundamental knowledge from classes, which, although it may take time, leads to a deep and permanent understanding of various processes that can be applied to other problems in the future.

(e) Sufficient experience in using the software The ability to use software efficiently can indeed depend on the time spent practicing in front of a computer screen. The more time spent practicing in front of the computer screen, the more experience one gains. These experiences cannot be acquired merely by sitting and listening in class but arise from hands-on practice. Anything that is attempted and does not produce the desired flow solutions becomes a valuable lesson for solving other flow problems in the future.

From the five components of knowledge as described above, it is understandable that solving flow problems by CFD is not just about having knowledge in the finite element method or finite volume method alone. It requires a comprehensive understanding of various fundamental areas, which must be accumulated over a certain period. This knowledge is not only applicable to solving flow problems but also opens opportunities to solve many other engineering and scientific problems governed by the differential equations, under different boundary conditions, and with complicated geometries.

1.4 Conclusion

Computational fluid dynamics (CFD) has become significantly important in both the study and design of various engineering tasks today. From calculating the flow conditions around airplanes, cars, or trucks, to analyzing the lift and drag forces of aircrafts, the flow through water pump impellers, the air flow and temperature levels in air-conditioned rooms or large buildings, predicting weather conditions over large areas, or simply controlling the air flow through microchips to reduce the temperature of small electronic devices. The benefits of solving fluid flow problems computationally have led to extensive research and development. This, in turn, has resulted in sophisticated commercial software available at high prices.

This chapter broadly discusses three main factors that lead to different outcomes, whether it be problems of flow, solids, heat transfer, or others. The results depend on: (a) the differential equations that describe the fact of the problem, (b) the boundary conditions, and (c) the geometry of the problem. In practice, these three factors are inherently complex, such as the set of the Navier-Stokes differential equations and complex boundary conditions in flow problems, combined with the complicated geometries of the problem. This complexity makes it impossible to find exact solutions in the same way as one might have found in certain academic courses. This is a reason why students may not fully realize the potential and the deep benefits of fluid mechanics courses as they ought to.

Therefore, solving complicated fluid flow problems using numerical methods on the computers is the only way to produce meaningful and valuable practical results. However, the process leading to such results requires the analyst to possess knowledge components in various areas. This can be categorized into five main parts: (a) understanding of the governing differential equations and their physical significance, (b) knowledge of the numerical methods necessary for computation, (c) knowledge in finite element method or finite volume method, which can model the complex shapes of problems, (d) a brief understanding of computer programs to help improve the efficiency of calculations and confidence in the accuracy of the solutions obtained, and (e) experience in using computer software, which depends on self-practice. These five components not only enable the analyst to solve fluid flow problems effectively but also open opportunities to analyze other engineering and scientific problems as well.

Chapter 2

The Navier-Stokes Equations

2.1 Introduction

Analyzing any problem in engineering or science, the nature of the results obtained as described in Chapter 1 typically depends on three main components which are the system of differential equations, the boundary conditions, and the geometry of the problem. For solving computational fluid dynamics problems, understanding the system of differential equations of flow is crucial. These differential equations describe the realities of flow which must conserve mass, momentums, and energy. These equations are composed of terms in derivative forms. Therefore, it is essential to understand and be familiar with these systems of differential equations, not just viewing them in the mathematical equation aspect, but also understanding the physical significance of them. Such understanding not only leads to successful problem-solving but also results in accurate, verifiable solutions, leading to a better understanding of the problem.

To easily understand the physical meaning of terms in the system of differential equations and their origins, this chapter will show the steps to derive the equations for two-dimensional flow. This significantly reduces the length of the equations as compared to three-dimensional flow. However, the understanding gained can be extended to three-dimensional flow directly. The terms in these equations are in the form of derivatives, which might not seem familiar when starting to study. However, understanding the physical meaning of these terms is necessary and crucial for solving CFD problems. The accuracy and precision of the computed solutions heavily rely on a deep understanding of these systems of differential equations.

2.2 Substantial Derivative

The substantial derivative is essential to understand before starting to develop a system of differential equations for fluid flow. To explain the meaning of such derivative, Figure 2.1 illustrates the path of an infinitesimally small fluid element moving from position 1 at time t_1 to position 2 at time t_2 on the x-y plane. On this plane, $\hat{i}$ and $\hat{j}$ are the unit vectors in the x and y-direction, respectively. The fluid element has a velocity $\vec{V}_1$ at position 1 and $\vec{V}_2$ at position 2. These velocities are the vector quantities, which can be expressed as,

$$\vec{V} = u\hat{i} + v\hat{j} \tag{2.1}$$

where u and v are the velocity components in the x and y-direction, respectively. These velocity components u and v depend on the coordinates x, y and time t as,

$$u = u(x, y, t) \tag{2.2a}$$

$$v = v(x, y, t) \tag{2.2b}$$

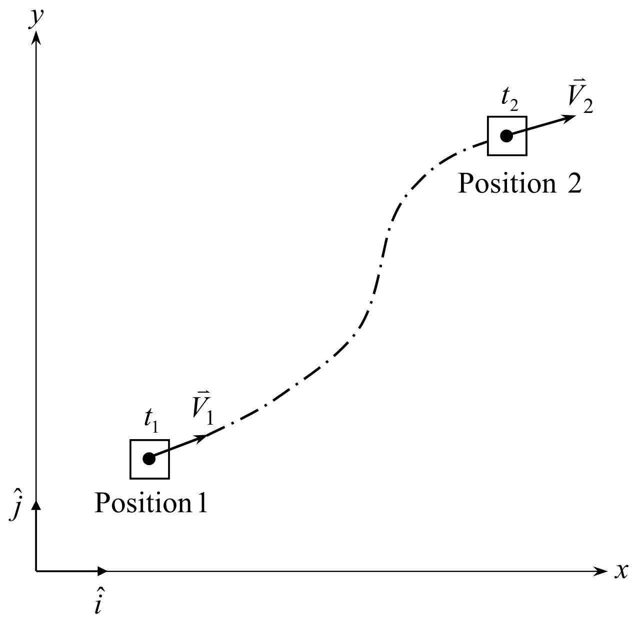


Figure 2.1 Flow path of a fluid element from position 1 at time t_1 to position 2 at time t_2.

As this fluid element moves, its properties change depending on the coordinate position and time. For example, if we consider the fluid density,

$$\rho = \rho(x, y, t) \tag{2.3a}$$

The density of the fluid element at position 1 is,

$$\rho_1 = \rho(x_1, y_1, t_1) \tag{2.3b}$$

As the fluid element moves to position 2, its density is,

$$\rho_2 = \rho(x_2, y_2, t_2) \tag{2.3c}$$

Since the change is continuous, the density at position 2 can be expressed in terms of the density at position 1 by using the Taylor series as,

$$\rho_2 = \rho_1 + \left(\frac{\partial \rho}{\partial x}\right)_1 \Delta x + \left(\frac{\partial \rho}{\partial y}\right)_1 \Delta y + \left(\frac{\partial \rho}{\partial t}\right)_1 \Delta t + H.O.T. \tag{2.4}$$

where $H.O.T.$ denotes the Higher Order Terms containing the higher orders of $\Delta x,$ Δy and Δt. By moving ρ_1 from the right to the left side of the equation and divide through by Δt, we obtain,

$$\frac{\rho_2 - \rho_1}{\Delta t} = \left(\frac{\partial \rho}{\partial x}\right)_1 \frac{\Delta x}{\Delta t} + \left(\frac{\partial \rho}{\partial y}\right)_1 \frac{\Delta y}{\Delta t} + \left(\frac{\partial \rho}{\partial t}\right)_1 + H.O.T. \tag{2.5}$$

As $\Delta t \to 0$, i.e., position 2 approaches position 1, terms in Eq. (2.5) become,

$$\lim_{\Delta t \to 0} \frac{\rho_2 - \rho_1}{\Delta t} = \frac{D\rho}{Dt} \tag{2.6}$$

$$\lim_{\Delta t \to 0} \frac{\Delta x}{\Delta t} = u \tag{2.7}$$

$$\lim_{\Delta t \to 0} \frac{\Delta y}{\Delta t} = v \tag{2.8}$$

and the terms in $H.O.T.$ containing $\Delta x, \Delta y$ and Δt all approach zero. Hence, Eq. (2.5) becomes,

$$\frac{D\rho}{Dt} = u\frac{\partial \rho}{\partial x} + v\frac{\partial \rho}{\partial y} + \frac{\partial \rho}{\partial t} \tag{2.9}$$

Each term in Eq. (2.9) has its physical meaning. The derivative term $D\rho/Dt$ on the left side of the equation represents the rate of change of density as a fluid element passes through position 1. That is, an observer follows this fluid element and observes the overall change in density within it. This term, which has such a physical meaning, is in the form of the *substantial derivative* $D\rho/Dt$, which defines the absolute change throughout the flow past any position. The substantial derivative $D\rho/Dt$ like this differs in meaning from the *local derivative* $\partial\rho/\partial t$, which is the last term on the right side of Eq. (2.9). The local derivative $\partial\rho/\partial t$ indicates the rate of change of density of any fluid element passing through position 1. That is, this time the observer remains stationary at position 1 to observe the changes in density of various fluid elements passing through. From this description, it is evident that the derivative terms D/Dt and $\partial/\partial t$ have different physical meanings and values, as clearly seen from Eq. (2.9).

The relationship as shown in Eq. (2.9) can be applied to other flow properties. Therefore, this relationship's equation can generally be written as follows,

$$\frac{D}{Dt} \equiv \frac{\partial}{\partial t} + u\frac{\partial}{\partial x} + v\frac{\partial}{\partial y} \tag{2.10}$$

Or,

$$\frac{D}{Dt} \equiv \frac{\partial}{\partial t} + \left(\vec{V} \cdot \vec{\nabla}\right) \tag{2.11}$$

where

$$\vec{\nabla} \equiv \frac{\partial}{\partial x}\hat{i} + \frac{\partial}{\partial y}\hat{j} \tag{2.12}$$

Equation (2.11) can further emphasize the understanding of the explanation in the previous paragraph. The term at the end on the right side of this equation is,

$$\begin{aligned}\left(\vec{V} \cdot \vec{\nabla}\right) &= \left(u\hat{i} + v\hat{j}\right)\cdot\left(\frac{\partial}{\partial x}\hat{i} + \frac{\partial}{\partial y}\hat{j}\right) \\ &= u\frac{\partial}{\partial x} + v\frac{\partial}{\partial y}\end{aligned} \tag{2.13}$$

which consists of two terms that have the velocity component in each direction multiplied in front. These two terms are called the flow *convection derivatives* since they are the product of the flow velocity components and derivatives of other flow properties. We will find that these terms will create difficulty in solving fluid mechanics problems. The first noticeable role at this preliminary stage is that the coefficients of this term include velocity components u and v, which are unknown variables in fluid flow solutions. Therefore, when such terms appear in any equation, that equation becomes nonlinear, directly leading to the complexity and difficulty in solving the problem by any numerical method.

When we observe a fluid element moving continuously, the properties of this fluid element change constantly as well. This includes the values of density ρ, velocity u and v, and temperature T, leading to the corresponding substantial derivatives $D\rho/Dt$, Du/Dt, Dv/Dt and DT/Dt, respectively. For example, the substantial derivative of the temperature,

$$\frac{DT}{Dt} = \frac{\partial T}{\partial t} + \left(\vec{V} \cdot \vec{\nabla} T\right) \tag{2.14}$$

$$\frac{DT}{Dt} = \frac{\partial T}{\partial t} + \left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) \tag{2.15}$$

This means the total change in temperature (the left-hand side term) of a fluid element we are observing as it moves, consists of the change in temperature at a specific location (the first right-hand side term) plus the change in temperature due to convection as the fluid element moves through that location (the terms in bracket on the right-hand side). Equation (2.15) can be further explained to create a clearer picture by using the following example. Suppose we are walking through a door into an air-conditioned room, we will feel cooler. This sensation is the derivative term of convection (convective term in the right-hand bracket), due to walking from a warmer place through the door at speeds u and v into a cooler air-conditioned room. At the same time, at the position of the door, there are cool water droplets spraying onto us, causing a further decrease in temperature (the first right-hand side term). Therefore, the overall sensation of cooling (the left-hand side term) includes the change in temperature as we walk into the cooler room combined with the coolness from the water droplets sprayed at the door.

Equation (2.14) is a commonly encountered equation in various subjects related to differential equations at the undergraduate level. The temperature of a fluid element depends on its coordinate position and time,

$$T = T(x, y, t) \tag{2.16}$$

If we apply the chain rule, we obtain,

$$dT = \frac{\partial T}{\partial x}dx + \frac{\partial T}{\partial y}dy + \frac{\partial T}{\partial t}dt \tag{2.17}$$

and divide through by dt to obtain, ,

$$\frac{dT}{dt} = \frac{\partial T}{\partial x}\frac{dx}{dt} + \frac{\partial T}{\partial y}\frac{dy}{dt} + \frac{\partial T}{\partial t} \tag{2.18}$$

Since $dx/dt = u$ and $dy/dt = v$, then,

$$\frac{dT}{dt} = \frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} \tag{2.19}$$

The term dT/dt on the left side of this equation is called the total derivative, which has the same meaning as the substantial derivative mentioned above because the terms on the right side of Eqs. (2.19) and (2.15) are the same. However, the explanation of the substantial derivative in Eq. (2.15) leads to a more understandable physical interpretation, which will lead to a better understanding of various terms in the Navier-Stokes equations, including how to deal with them in the computational process.

2.3 Conservation of Mass

The equations in the system of partial differential equations for fluid flow need to represent physical realities that are meaningful and easily understandable. The first equation in this fluid flow partial differential equation system is the conservation of mass equation, which can be simply understood as the mass is conserved. The derivation of this conservation equation can be done easily by considering the flow through a small element with dimensions of *dx* and *dy*, and a depth of one unit, at any given fixed position in the flow domain as illustrated in Figure 2.2.

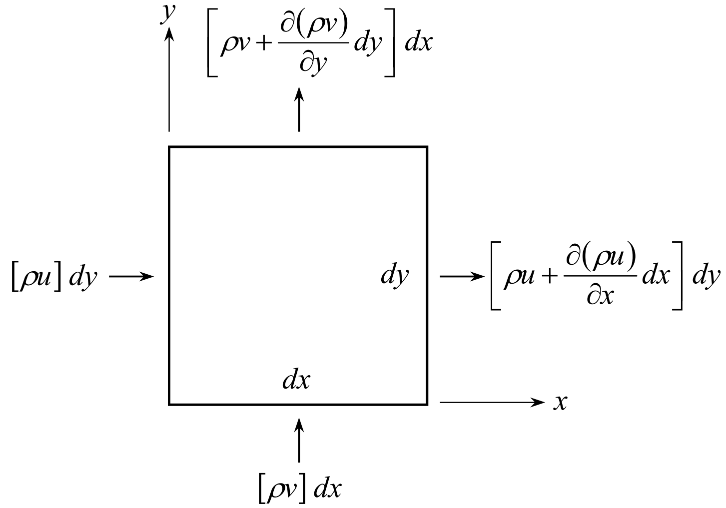


Figure 2.2 Mass fluxes through a small element fixed in the flow domain for constructing the conservation of mass equation.

Along the left edge *dy* of this small element, the mass flux entering is $[\rho u]dy$. Since the density ρ and the velocity u changing continuously, therefore, the mass flux exiting on the right edge of the frame is $[\rho u + (\partial(\rho u)/\partial x)dx]dy$. That is, the increase in mass flux in the x-direction through the *dy* edge of the flow through this small frame is,

$$\left[\rho u + \frac{\partial(\rho u)}{\partial x} dx\right] dy - [\rho u] dy = \frac{\partial(\rho u)}{\partial x} dx\, dy \tag{2.20}$$

Similarly, the mass flux through the edge *dx* from the bottom to the top of the element is,

$$\left[\rho v + \frac{\partial(\rho v)}{\partial y} dy\right] dx - [\rho v] dx = \frac{\partial(\rho v)}{\partial y} dx\, dy \tag{2.21}$$

Since the mass of this small element is $\rho\, dx\, dy$, therefore, the rate of change of mass or the decrease in mass flux is,

$$-\frac{\partial \rho}{\partial t} dx\, dy \tag{2.22}$$

But because the mass in this small element must be conserved, it means that the increase in mass flux from the flow through the edges dx and dy must equal the decrease in mass flux within this small element, i.e.,

$$\frac{\partial(\rho u)}{\partial x} dx\, dy + \frac{\partial(\rho v)}{\partial y} dx\, dy = -\frac{\partial \rho}{\partial t} dx\, dy$$

Dividing through by $dx\, dy$ and arrange the equation to obtain,

$$\frac{\partial \rho}{\partial t} + \left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y}\right] = 0 \tag{2.23}$$

Or,
$$\frac{\partial \rho}{\partial t} + \vec{\nabla}\cdot\left(\rho \vec{V}\right) = 0 \tag{2.24}$$

This Eq. (2.24) is the conservation of mass equation, which is the first equation in the system of partial differential equations for fluid flow. It indicates that mass is not lost. This equation is in the form of first-order derivatives, involving three unknowns of ρ, u, and v, which can change throughout the flow domain. Therefore, this single mass conservation equation cannot be used to solve the problem alone. It is necessary to develop additional equations so that the flow behavior can be solved.

2.4 Conservation of Momentums

The second type of physical reality of flow that can be formulated into additional partial differential equations comes from applying the Newton's second law. The law states that the force equals mass times acceleration. In applying the Newton's second law, we consider a mass element with dimensions of dx and dy, and a depth of one unit as shown in Figure 2.3, which is moving with the flow.

Figure 2.3 illustrates only the forces acting in the x-direction. The Newton's second law in the x-direction is,

$$F_x = m a_x \tag{2.25}$$

where F_x is the total force in the x-direction, m is the mass of the element, and a_x is the acceleration of the element in the x-direction.

The total force in the x-direction consists of forces acting on edges and the body force. The forces acting on edges include forces due to pressure p, normal stress, and shear stress. For shear stress, the first subscript (y in this case) specifies the edge perpendicular to the y-axis, where this shear stress is acting. The second subscript (x in this case) indicates the direction of the shear stress. Therefore, the total force acting on edges in the x-direction of this element is,

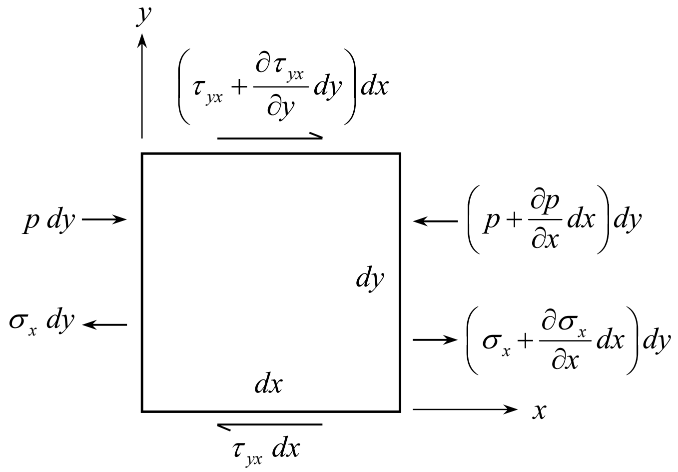


Figure 2.3 Forces in the x-direction acting on an element of fluid moving with the flow used in formulating the conservation of momentum equation.

$$\left[p - \left(p + \frac{\partial p}{\partial x} dx\right)\right] dy + \left[\left(\sigma_x + \frac{\partial \sigma_x}{\partial x} dx\right) - \sigma_x\right] dy + \left[\left(\tau_{yx} + \frac{\partial \tau_{yx}}{\partial y} dy\right) - \tau_{yx}\right] dx \tag{2.26}$$

The body force of the element in the x-direction is,

$$\rho f_x (dx\,dy) \tag{2.27}$$

Thus the total force in the x-direction from Eqs. (2.26) and (2.27) is,

$$F_x = \left[-\frac{\partial p}{\partial x} + \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y}\right] dx\,dy + \rho f_x\,dx\,dy \tag{2.28}$$

while the mass of this element is,

$$m = \rho(dx\,dy) \tag{2.29}$$

The acceleration a_x of the element in Eq. (2.25) is the rate of change of the velocity u with respect to time t. Since we are observing this element as it moves with the flow, this acceleration is the total derivative of u, that is,

$$a_x = \frac{Du}{Dt} \tag{2.30}$$

By substituting Eqs. (2.28)-(2.30) into the Newton's second law in Eq.(2.25) and dividing it through by *dx dy*, we obtain,

$$\rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \rho f_x \tag{2.31a}$$

Similarly, application of the Newton's second law in the y-direction leads to,

$$\rho\frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \rho f_y \tag{2.31b}$$

Equations (2.31a-b) are called the Navier- Stokes equations, in honor of the Frenchman named M. Navier and the Englishman named G. Stokes, who independently derived these equations.

The Navier-Stokes equations (2.31a-b) are in the form of the substantial derivatives resulting from the formulation of the equations by observing a moving fluid element. This substantial derivative can be transformed into the form of the local or partial derivatives, which is akin to an observer standing still at a certain position and watching the changes of the moving fluid. This is done by applying the relationship in Eq. (2.11) to the velocity u as follows,

$$\frac{Du}{Dt} = \frac{\partial u}{\partial t} + \vec{V}\cdot\vec{\nabla}u \tag{2.32}$$

Then,
$$\rho\frac{Du}{Dt} = \rho\frac{\partial u}{\partial t} + \rho\vec{V}\cdot\vec{\nabla}u \tag{2.33}$$

That is, the term on the left-hand side of the Navier-Stokes equations in the form of substantial derivatives can be represented by both terms on the right-hand side of equation (2.33), which are in the form of the local or partial derivatives, and can be used in conjunction with the conservation of mass Eq. (2.24). This is because every term in these equations is already in the form of partial derivatives. Note that the two terms on the right-hand side of Eq. (2.33) can be simplified by using the following relationships.

Since,
$$\frac{\partial(\rho u)}{\partial t} = \rho\frac{\partial u}{\partial t} + u\frac{\partial \rho}{\partial t}$$

Then,
$$\rho\frac{\partial u}{\partial t} = \frac{\partial(\rho u)}{\partial t} - u\frac{\partial \rho}{\partial t} \tag{2.34}$$

Also, since $$\vec{\nabla}\cdot\left(\rho u\vec{V}\right) = u\vec{\nabla}\cdot(\rho\vec{V}) + (\rho\vec{V})\cdot\vec{\nabla}u$$

Then, $$\rho\vec{V}\cdot\vec{\nabla}u = \vec{\nabla}\cdot(\rho u\vec{V}) - u\vec{\nabla}\cdot(\rho\vec{V}) \tag{2.35}$$

By substituting Eqs. (2.34)-(2.35) into Eq. (2.33), we obtain,

$$\rho\frac{Du}{Dt} = \frac{\partial(\rho u)}{\partial t} - u\frac{\partial\rho}{\partial t} + \vec{\nabla}\cdot\left(\rho u\vec{V}\right) - u\vec{\nabla}\cdot\left(\rho\vec{V}\right)$$

$$\rho\frac{Du}{Dt} = \frac{\partial(\rho u)}{\partial t} - u\left[\frac{\partial\rho}{\partial t}+\vec{\nabla}\cdot\left(\rho\vec{V}\right)\right] + \vec{\nabla}\cdot\left(\rho u\vec{V}\right) \tag{2.36}$$

Since the sum of the two terms in the square bracket equals zero according to the conservation of mass Eq. (2.24), then Eq. (2.36) becomes,

$$\rho\frac{Du}{Dt} = \frac{\partial(\rho u)}{\partial t} + \vec{\nabla}\cdot\left(\rho u\vec{V}\right) \tag{2.37}$$

Substitute Eq. (2.37) into Eq. (2.31a) to obtain,

$$\frac{\partial(\rho u)}{\partial t} + \vec{\nabla}\cdot\left(\rho u\vec{V}\right) = -\frac{\partial p}{\partial x} + \frac{\partial\sigma_x}{\partial x} + \frac{\partial\tau_{yx}}{\partial y} + \rho f_x \tag{2.38a}$$

Similarly, Eq. (2.31b) can be written as,

$$\frac{\partial(\rho v)}{\partial t} + \vec{\nabla}\cdot\left(\rho v\vec{V}\right) = -\frac{\partial p}{\partial y} + \frac{\partial\tau_{xy}}{\partial x} + \frac{\partial\sigma_y}{\partial y} + \rho f_y \tag{2.38b}$$

Equations (2.38a-b) are known as the Navier-Stokes equations in the conservation form.

The terms on the right-hand side of the Navier-Stokes equations consist of normal stresses and shear stress, which must be written in terms of the velocities along the x- and y-coordinates. In the late 17th century, Sir Isaac Newton proposed that these stress components vary directly with the change in velocity (velocity gradient). This proposal was found to be applicable to fluids in general, leading to its acceptance and calling fluids with such properties as Newtonian fluids. This proposal led to the establishment of a relationship between stress and velocity components as follows,

$$\sigma_x = \lambda\left(\vec{\nabla}\cdot\vec{V}\right) + 2\mu\frac{\partial u}{\partial x} \tag{2.39a}$$

$$\sigma_y = \lambda\left(\vec{\nabla}\cdot\vec{V}\right) + 2\mu\frac{\partial v}{\partial y} \tag{2.39b}$$

and $$\tau_{xy} = \tau_{yx} = \mu\left[\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right] \tag{2.39c}$$

where μ is the dynamic viscosity or sometimes called the first viscosity, and λ is the second viscosity. Stokes proposed that,

$$\lambda = -\frac{2}{3}\mu \tag{2.40}$$

The above relation is known as the Stokes's hypothesis, and has been found that it works well when the fluid is a gas. However, for liquids, the fluid density ρ for most of the flow problems remains constant, the conservation of mass Eq. (2.4) simplifies to $\vec{\nabla}\cdot\vec{V} = 0$ or $div\,\vec{V} = 0$. This results in the second viscosity λ not being used in calculations, and the normal stress components in Eqs. (2.39a-b) becomes twice the product of the dynamic viscosity and the velocity gradient. For this reason, research into the true value of the second viscosity λ has not been extensively pursued, leading to a lack of clarity and definitive confirmation up to the present day.

By substituting the stress components, which are in terms of the velocity components, from Eqs. (2.39a-c) into Eqs. (2.38a-b), it results in the Navier-Stokes equations in conservation form as,

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial\left(\rho u^2\right)}{\partial x} + \frac{\partial(\rho u v)}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(\lambda\vec{\nabla}\cdot\vec{V} + 2\mu\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + \rho f_x \tag{2.41a}$$

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho u v)}{\partial x} + \frac{\partial\left(\rho v^2\right)}{\partial y} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + \frac{\partial}{\partial y}\left(\lambda\vec{\nabla}\cdot\vec{V} + 2\mu\frac{\partial v}{\partial y}\right) + \rho f_y \tag{2.41b}$$

Equations (2.41a-b) illustrate the complexity of the Navier- Stokes equations, where the terms are in the form of derivatives of unknown variables. Moreover, the second and third terms on the left- hand side of both these equations are nonlinear terms, which add to the complexity of solving such a system of partial differential equations, regardless of the numerical methods used.

2.5 Conservation of Energy

For many types of flow, such as the flow around spacecraft moving at several times the speed of sound to the flow of hot air under a roof, the velocity of the flow and the changing temperature within the fluid depend on each other. Therefore, the third fundamental truth of any flow that can be used to formulate an additional partial differential equation is the law of conservation of energy. Figure 2.4 illustrates a fluid element with dimensions of *dx* and *dy*, having a depth of one unit, which is moving with the flow.

The conservation of energy equation can be formulated using the first law of thermodynamics, which states that the rate of energy change in a fluid element is equal to the heat flux supplied to the element plus the rate of work done due to forces acting on that element, that is,

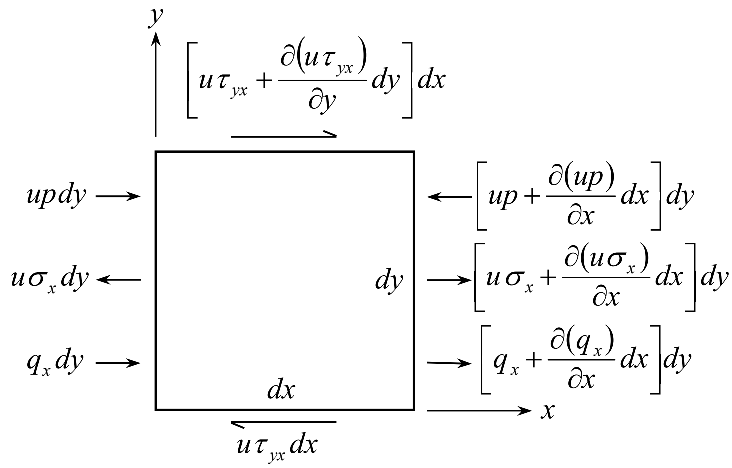


Figure 2.4 Work done and flux quantities in the *x*-direction acting on an element of fluid moving with the flow used in formulating the conservation of energy equation.

Rate of energy change in fluid element	=	Heat flux supplied to the element	+	Rate of work done due to forces acting on that element	
Or, A	=	B	+	C	(2.42)

where the terms A, B and C have their physical meanings as stated in Eq. (2.42).

If we begin by considering term C, which represents the rate of work resulting from forces acting on this element, the first type of force is the force from the weight of the element itself, which, when multiplied by the speed of the flow in that direction, results in the rate of work as,

$$\rho\vec{f}\cdot\vec{V}\,(dx\,dy)$$

From Figure 2.4, the rate of work resulting from the pressure p acting on the side dy in the x-direction is,

$$\left[up - \left(up + \frac{\partial(up)}{\partial x}dx\right)\right]dy \quad = \quad -\frac{\partial(up)}{\partial x}dx\,dy$$

The rate of work resulting from the normal stress σ_x acting on the side *dy* in the *x*-direction is,

$$\left[u\sigma_x + \frac{\partial(u\sigma_x)}{\partial x}dx\right]dy - u\sigma_x\,dy \quad = \quad \frac{\partial(u\sigma_x)}{\partial x}dx\,dy$$

The rate of work resulting from the shear stress τ_{yx} acting on the side *dx* in the *x*-direction is,

$$\left[u\tau_{yx} + \frac{\partial(u\tau_{yx})}{\partial y}dy\right]dx - u\tau_{yx}\,dx \quad = \quad \frac{\partial(u\tau_{yx})}{\partial y}dx\,dy$$

Similarly, the rate of work resulting from forces acting on the mass in the *y*-direction can also be derived, leading to the total rate of work arising from the forces on this mass as,

$$C \;=\; \left[-\left(\frac{\partial(up)}{\partial x} + \frac{\partial(vp)}{\partial y}\right) + \frac{\partial(u\sigma_x)}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\sigma_y)}{\partial y}\right]dx\,dy + \rho\vec{f}\cdot\vec{V}\,dx\,dy \qquad (2.43)$$

For term *B*, which represents the heat flux supplied to the element, it consists of two parts. The first part is the heat flux occurring within the volume of the element. For example, the heat flux generated internally within the element which is normally called the internal heat generation. If the amount of heat flux generated per unit mass is $\overline{Q}$, then the amount of internal heat flux for this element is,

$$\rho\overline{Q}(dx\,dy)$$

From Figure 2.4, the net heat flux resulting from heat transfer in the *x*- direction through the edge *dy* on both on the left and right edges of the element is,

$$\left[q_x - \left(q_x + \frac{\partial q_x}{\partial x}dx\right)\right]dy \quad = \quad -\frac{\partial q_x}{\partial x}dx\,dy$$

Similarly, the net heat flux resulting from heat transfer in the *y*-direction through the edge *dx* on both at the bottom and top edges of the element is,

$$\left[q_y - \left(q_y + \frac{\partial q_y}{\partial y}dy\right)\right]dx \quad = \quad -\frac{\partial q_y}{\partial y}dx\,dy$$

Therefore, the total heat flux on this element is,

$$B = \left[\rho\bar{Q} - \frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y} \right] dx\, dy \tag{2.44}$$

But according to the Fourier's law, the amount of heat fluxes q_x and q_y that vary with the temperature gradient are,

$$q_x = -k\frac{\partial T}{\partial x} \quad \text{and} \quad q_y = -k\frac{\partial T}{\partial y} \tag{2.45}$$

where k is the fluid thermal conductivity coefficient. Then, term B becomes,

$$B = \left[\rho\bar{Q} + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) \right] dx\, dy \tag{2.46}$$

Term A, which represents the rate of change of energy within the element consisting of the internal and kinetic energy. The internal energy arises from the movement of molecules within the fluid, while the kinetic energy occurs from the fluid's motion. If e represents the internal energy and $V^2/2$ represents the kinetic energy of the element flowing at speed V, then the total energy, which has units per unit mass, is $e+V^2/2$. Since the total mass of this element is $\rho\, dx\, dy$, therefore, term A is,

$$A = \rho\frac{D}{Dt}\left(e+\frac{V^2}{2}\right) dx\, dy \tag{2.47}$$

By substituting term A, which is the rate of change of energy in the element from Eq. (2.47), and term B, which is the heat flux provided to the element from Eq. (2.46), along with term C, which is the rate of work resulting from various forces on the element from Eq. (2.43), into Eq. (2.42) and then dividing throughout by $dx\, dy$, it results in the conservation of energy equation as,

$$\rho\frac{D}{Dt}\left(e+\frac{V^2}{2}\right) = \rho\bar{Q} + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) - \frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y}$$
$$+ \frac{\partial(u\sigma_x)}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\sigma_y)}{\partial y} + \rho\vec{f}\cdot\vec{V} \tag{2.48}$$

The derived conservation of energy Eq. (2.48) is in the form of the substantial derivative, which needs to be converted into the form of local derivatives to be used in conjunction with the mass conservation equation (2.24) and the momentum conservation equation (2.41). The substantial derivative in the energy conservation equation (2.48) acts on both the internal energy term e and the kinetic energy term. Therefore, to simplify the derivation process, let's first demonstrate the steps to convert the substantial to local derivative of only the internal energy e, as follows.

The steps to convert such derivative form start from multiplying Eqs. (2.31a) and (2.31b) by the velocities *u* and *v*, respectively, as,

$$\rho\frac{D(u^2/2)}{Dt} = -u\frac{\partial p}{\partial x} + u\frac{\partial \sigma_x}{\partial x} + u\frac{\partial \tau_{yx}}{\partial y} + \rho u f_x \tag{2.49a}$$

$$\rho\frac{D(v^2/2)}{Dt} = -v\frac{\partial p}{\partial y} + v\frac{\partial \tau_{xy}}{\partial x} + v\frac{\partial \sigma_y}{\partial y} + \rho v f_y \tag{2.49b}$$

By combining the above two equations together and using the relation, $u^2+v^2 = V^2$, we obtain,

$$\rho\frac{D(V^2/2)}{Dt} = -u\frac{\partial p}{\partial x} - v\frac{\partial p}{\partial y} + u\left(\frac{\partial \sigma_x}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}\right) + v\left(\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \sigma_y}{\partial y}\right) + \rho(uf_x+vf_x) \tag{2.50}$$

Then, subtracting Eq. (2.50) from Eq. (2.48) and using the relation, $\rho\vec{f}\cdot\vec{V} = \rho\left(uf_x+vf_y\right)$, this leads to,

$$\rho\frac{De}{Dt} = \rho\bar{Q} + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) - p\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right) + \sigma_x\frac{\partial u}{\partial x} + \tau_{yx}\frac{\partial u}{\partial y} + \tau_{xy}\frac{\partial v}{\partial x} + \sigma_y\frac{\partial v}{\partial y} \tag{2.51}$$

The terms on the left-hand side of the above equation consist only the substantial derivative acting on the internal energy *e*. The terms on the right-hand side this equation are less complex than those in the energy conservation Eq. (2.48), which includes the derivatives involving velocities and stresses multiplied together, as well as the inclusion of body forces. Equation (2.51) can be further simplified because τ_{xy} must equal τ_{yx}, otherwise the small mass element in Figure 2.4 would rotate about itself. Therefore, Eq. (2.51) becomes,

$$\rho\frac{De}{Dt} = \rho\bar{Q} + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) - p\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right) + \sigma_x\frac{\partial u}{\partial x} + \sigma_y\frac{\partial v}{\partial y} + \tau_{yx}\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right) \tag{2.52}$$

Then, by substituting the stress components, Eq. (2.52), in form of the velocity components through Eqs. (2.39a)-(2.39c), we obtain,

$$\rho\frac{De}{Dt} = \rho\bar{Q} + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) - p\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right) + \lambda\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)^2 + \mu\left[2\left(\frac{\partial u}{\partial x}\right)^2 + 2\left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2\right] \tag{2.53}$$

The substantial derivative of the internal energy can be transformed into the form of local derivatives. This can be done by starting from the use of the definition of the substantial derivative in Eq. (2.11), applied to the internal energy value *e*, and then multiplied throughout by the density ρ,

$$\rho \frac{De}{Dt} = \rho \frac{\partial e}{\partial t} + \rho \vec{V} \cdot \vec{\nabla} e \tag{2.54}$$

Since,

$$\frac{\partial(\rho e)}{\partial t} = \rho \frac{\partial e}{\partial t} + e \frac{\partial \rho}{\partial t}$$

then,

$$\rho \frac{\partial e}{\partial t} = \frac{\partial(\rho e)}{\partial t} - e \frac{\partial \rho}{\partial t} \tag{2.55}$$

In addition, by applying the divergence theorem onto the product of a scalar and a vector,

$$\vec{\nabla} \cdot \left(\rho e \vec{V}\right) = e \vec{\nabla} \cdot \left(\rho \vec{V}\right) + \left(\rho \vec{V}\right) \cdot \vec{\nabla} e$$

Or,

$$\rho \vec{V} \cdot \vec{\nabla} e = \vec{\nabla} \cdot \left(\rho e \vec{V}\right) - e \vec{\nabla} \cdot \left(\rho \vec{V}\right) \tag{2.56}$$

By substituting Eq. (2.55) and (2.56) into Eq. (2.54) and arranging terms to obtain,

$$\rho \frac{De}{Dt} = \frac{\partial(\rho e)}{\partial t} - e\left[\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot \left(\rho \vec{V}\right)\right] + \vec{\nabla} \cdot \left(\rho e \vec{V}\right)$$

It is noted that the summation within the square bracket is zero according to the conservation of mass, Eq. (2.24), thus,

$$\rho \frac{De}{Dt} = \frac{\partial(\rho e)}{\partial t} + \vec{\nabla} \cdot \left(\rho e \vec{V}\right) \tag{2.57}$$

Equations (2.57) is then substituted into the left-hand side of Eq. (2.53) to yield,

$$\frac{\partial(\rho e)}{\partial t} + \vec{\nabla} \cdot \left(\rho e \vec{V}\right) = \rho \bar{Q} + \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k \frac{\partial T}{\partial y}\right) - p\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)$$
$$+ \lambda\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)^2 + \mu\left[2\left(\frac{\partial u}{\partial x}\right)^2 + 2\left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2\right] \tag{2.58}$$

which is the conservation of energy equation written only in form of the internal energy *e*.

However, since the rate of change of the total energy within an element consists of the internal energy *e* and the kinetic energy $V^2/2$, the absolute derivative term on the left-hand side of Eq. (2.48) must be written in the form of the local derivative. This is done by following the steps from Eq. (2.54) to Eq. (2.57), with the substitution of the internal energy term *e* by the total energy term $e + V^2/2$ to obtain,

$$\rho\frac{D\left(e+V^2/2\right)}{Dt} \quad = \quad \frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right] \;+\; \vec{\nabla}\cdot\left[\rho\left(e+\frac{V^2}{2}\right)\vec{V}\right] \tag{2.59}$$

Finally, by substituting Eq. (2.59) into Eq. (2.48) leads to,

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right] \;+\; \vec{\nabla}\cdot\left[\rho\left(e+\frac{V^2}{2}\right)\vec{V}\right] \;=\; \rho\bar{Q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)$$

$$-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}+\frac{\partial(u\sigma_x)}{\partial x}+\frac{\partial\left(u\tau_{yx}\right)}{\partial y}+\frac{\partial\left(v\tau_{xy}\right)}{\partial x}+\frac{\partial\left(v\sigma_y\right)}{\partial y}+\rho\vec{f}\cdot\vec{V} \tag{2.60}$$

which is the conservation of energy equation written in term of the total energy. A detailed examination of this equation reveals that each term has its own mathematical complexity. Most terms are nonlinear, which contributes to the complexity in applying numerical methods. Specifically, several terms on the right-hand side are derivative terms of the product of velocity and stress components in different directions.

2.6 System of the Navier-Stokes Equations

The conservation of mass, momentum, and energy equations, which are derived from the fact that mass is not lost, the application of the Newton's second law, and energy is not lost, respectively, are elaborately detailed in sections 2.3-2.5. This leads to a system of the partial differential equations that can be summarized as follows.

Conservation of mass

$$\frac{\partial\rho}{\partial t} \;+\; \vec{\nabla}\cdot\left(\rho\vec{V}\right) \;=\; 0 \tag{2.61}$$

Conservation of momentums

x-dir:
$$\frac{\partial(\rho u)}{\partial t} \;+\; \vec{\nabla}\cdot\left(\rho u\vec{V}\right) \;=\; -\frac{\partial p}{\partial x}+\frac{\partial\sigma_x}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\rho f_x \tag{2.62a}$$

y-dir:
$$\frac{\partial(\rho v)}{\partial t} \;+\; \vec{\nabla}\cdot\left(\rho v\vec{V}\right) \;=\; -\frac{\partial p}{\partial y}+\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial\sigma_y}{\partial y}+\rho f_y \tag{2.62b}$$

Conservation of energy

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\vec{\nabla}\cdot\left[\rho\left(e+\frac{V^2}{2}\right)\vec{V}\right] = \rho\bar{Q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)-\frac{\partial(up)}{\partial x}$$

$$-\frac{\partial(vp)}{\partial y}+\frac{\partial(u\sigma_x)}{\partial x}+\frac{\partial\left(u\tau_{yx}\right)}{\partial y}+\frac{\partial\left(v\tau_{xy}\right)}{\partial x}+\frac{\partial\left(v\sigma_y\right)}{\partial y}+\rho\vec{f}\cdot\vec{V} \tag{2.63}$$

The four partial differential equations above are in conservation form. The observer keeps his eyes on an element of size *dx* and *dy* fixed in the flow domain, without moving with the flow, and monitor the flux quantities entering and leaving this element. A detailed examination of all four equations reveals that on the left side of each equation, there are terms related to the divergence of the flux quantity, i.e., $\vec{\nabla}\cdot$ (flux quantity) as follows.

Eq. (2.61):	$\rho\vec{V}$	is the mass flux
Eq. (2.62a):	$\rho u\vec{V}$	is the momentum flux in x-direction
Eq. (2.62b):	$\rho v\vec{V}$	is the momentum flux in y-direction
Eq. (2.63):	$\rho\left(e+\frac{V^2}{2}\right)\vec{V}$	is the total energy flux

Furthermore, upon closer inspection of all four equations, it is found that these four equations can be written in the same form, which is,

$$\frac{\partial\{U\}}{\partial t}+\frac{\partial\{E\}}{\partial x}+\frac{\partial\{F\}}{\partial y} = \{H\} \tag{2.64}$$

where $\{U\}$, $\{E\}$, $\{F\}$ and $\{H\}$ are vectors as follows.

$$\{U\} = \begin{Bmatrix} \rho \\ \rho u \\ \rho v \\ \rho\left(e+\frac{V^2}{2}\right) \end{Bmatrix} \tag{2.65}$$

$$\{E\} = \begin{Bmatrix} \rho u \\ \rho u^2+p-\sigma_x \\ \rho vu-\tau_{xy} \\ \rho\left(e+\frac{V^2}{2}\right)u+pu-k\frac{\partial T}{\partial x}-u\sigma_x-v\tau_{xy} \end{Bmatrix} \tag{2.66}$$

$$\{F\} = \begin{Bmatrix} \rho v \\ \rho uv-\tau_{yx} \\ \rho v^2+p-\sigma_y \\ \rho\left(e+\frac{V^2}{2}\right)v+pv-k\frac{\partial T}{\partial y}-u\tau_{yx}-v\sigma_y \end{Bmatrix} \tag{2.67}$$

$$\{H\} = \begin{Bmatrix} 0 \\ \rho f_x \\ \rho f_y \\ \rho\left(uf_x + vf_y\right) + \rho\bar{Q} \end{Bmatrix} \tag{2.68}$$

The advantage of writing all four partial differential equations in the same form, as shown in Eq. (2.64), is that in any process of applying numerical methods to the system, these equations can be viewed as essentially a single equation with similar characteristics, that is,

$$\frac{\partial U}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = H \tag{2.69}$$

This leads to the development of a generalized continuous equation resulting from the application of numerical methods, without concern for whether the equation is for conservation of mass, momentum, or energy. The derived continuous equation can be applied to any of the four partial differential equations, simplifying the process of formulating equations, enhancing confidence in their accuracy, and most importantly, facilitating the development of coherent computer programming. This significance becomes especially evident in the context of solving problems related to high-speed compressible flow, as discussed in Chapters 9 and 12.

However, regardless of the form in which the system of four partial differential equations comprising conservation of mass, momentums in the x and y-directions, and energy are written, these equations illustrate the significant difficulty of finding an exact solution through purely mathematical analysis. Even today, no exact solutions have been found for general flow problems governed by these four equations. This difficulty primarily arises from two main challenges. The first challenge is that these are coupled partial differential equations, where the results obtained, such as velocity components u, v, pressure p, and temperature T, must simultaneously satisfy all the four equations. The second challenge is that these equations are nonlinear, making it difficult to find exact solutions regardless of how simple the boundary conditions and geometry of the problem may be. These challenges have significantly contributed to the importance of Computational Fluid Dynamics (CFD). If one can solve this system of equations, a variety of results can be obtained, reflecting the characteristics of the flow and leading to a deeper understanding of the problem at hand. Additionally, it impresses upon those conducting calculations that sometimes, this set of four partial differential equations can produce remarkably diverse and complex flow behaviors.

As described in section 2.4, historically, the conservation of momentum Eqs. (2.41a-b) were referred to as the Navier-Stokes equations. However, in current computational fluid dynamics, this entire set of such partial differential equations, which includes the conservation of mass, momentums, and energy equations, has commonly come to be referred to as the system of the Navier-Stokes equations. This is because solving fluid dynamics problems requires addressing the entire set of these equations, and referring to them collectively as the Navier-Stokes equations simplifies communication and understanding, rather than having to separately mention the conservation of mass and energy equations.

When examining the four Navier-Stokes equations as presented in Eqs. (2.61)-(2.63), it's observed that these equations comprise five unknowns: ρ, u, v, p, and e. For incompressible flow, ρ is typically a constant and known beforehand, making the number of equations equal to the number of unknowns. However, for gas flows, ρ is not constant and becomes an unknown, especially in the case of compressible flow. Therefore, an additional equation is required. If the gas is considered a perfect gas, this additional equation could be the equation of state, such as,

$$p = \rho R T \tag{2.70}$$

where R is the universal gas constant. However, this equation introduces another unknown, which is the temperature T, necessitating the derivation of an additional equation. This new equation might be another type of state equation that illustrates the relationship between the temperature T and the internal energy e. For example,

$$e = c_v T \tag{2.71}$$

where c_v is the specific heat of the gas at constant volume, making the total number of equations equal to the number of unknowns, thereby the problem can be solved.

2.7 Boundary Conditions

All topics discussed in this chapter are related to the system of Navier-Stokes equations. The outcomes resulting from solving this system of Navier-Stokes equations depend on the boundary conditions and geometry of the problem. For example, in calculating the flow conditions in an air-conditioned room of a certain shape, if the cool air emitted from the air conditioner has different speeds, it will result in different flow conditions in the room, even though the same set of Navier-Stokes equations and the room's geometry remain unchanged.

At first glance, setting boundary conditions for flow problems may seem straightforward and uncomplicated, but in reality, regardless of the CFD method used, establishing correct and appropriate boundary conditions can be far from simple. In some cases, it can affect the construction of the domain size used in calculations, such as ensuring the domain is sufficiently large to make the boundary conditions along its edges as realistic as possible. Inappropriate or unrealistic boundary conditions can lead to significantly inaccurate results. Moreover, correctly setting appropriate boundary conditions depends on under-standing the type of Navier-Stokes equations under the flow conditions being studied, whether they are elliptic,