الجمهورية الجزائرية الديمقراطية الشعبية

The People's Democratic Republic of Algeria

وزارة التعليم العالي والبحث العلمي

Ministry of Higher Education and Scientific Research

جامعة عمار ثليجي – الأغواط

University Amar Telidji of Laghouat

Faculty : Technology

Department : : Mechanical Engineering

Domain : Technology

Field : Mechanical Engineering

Specialty : Energy

Course Handout (Lectures)

Intended for students of: Bachelor Level: 3rd Year

# Fluid MECHANICS 2


Presented by : Belkacem BOUALI

Maître de Conférences (A), University of Laghouat

Email : b.bouali@lagh-univ.dz


Academic year: 2024/2025.

# PREFACE

This document serves as the handout for the Fluid Mechanics 2 course, a crucial part of the third-year Bachelor's degree curriculum for Mechanical Engineering students specializing in Energy. Building upon the concepts introduced in Fluid Mechanics 1, this course delves deeper into the dynamic behavior of fluids, focusing on advanced topics essential for understanding and solving complex fluid-related problems encountered in engineering practice.

The course covers a wide range of topics, from fluid kinematics to the analysis of potential flows, and incorporates practical methods like finite control volume analysis and dimensional analysis. These methods form the foundation for modeling and solving real-world engineering systems such as jet engines, hydraulic systems, and rotating machinery.

By incorporating the most widely used fluid mechanics textbooks [1–6], which are referenced at the end of this document, this handout ensures that students are equipped with the theoretical knowledge and analytical tools necessary to tackle the complexities of fluid dynamics. I strongly encourage students to use these books as complementary resources, as they provide invaluable insight into both the mathematical foundations and practical applications of the subject.

Fluid mechanics is not just a theoretical discipline. It has profound implications in fields ranging from energy systems to aerodynamics. As you embark on this course, I hope you engage deeply with the material, applying the concepts learned here to real-world scenarios and engineering challenges.

I wish you success as you continue to build upon the solid foundation laid in Fluid Mechanics 1 and explore the fascinating world of fluid dynamics in this second part of the course.

**BOUALI Belkacem**
**Mechanical Engineering Department**
**Faculty of Technology**
**University of Amar Thelidji-Laghouat**

# CONTENTS

# INTRODUCTION

Fluid mechanics is a fundamental area of study in engineering that deals with the behavior and movement of fluids and their interactions with surfaces and boundaries. "**Fluid Mechanics 2**" serves as a continuation of the foundational concepts introduced in "**Fluid Mechanics 1**", where basic fluid properties and fluid statics were explored.

This document provides a comprehensive exploration of key fluid mechanics concepts, starting with fluid kinematics, which focuses on the motion of fluids without considering the forces causing that motion. We delve into the analysis of velocity and acceleration fields, along with streamlines, pathlines, and streaklines, to understand the behavior of fluid particles. Additionally, the document explores the differences between the Eulerian and Lagrangian methods of describing fluid flow, offering valuable insights into how fluid behavior can be observed and modeled.

Following this, the document introduces the concept of potential flow, which describes irrotational and incompressible fluid motion. By studying simple 2D potential flows such as uniform flow, source-sink flow, and free vortex flow, we explore how these basic flows can be superposed to create more complex flow patterns, such as Rankine ovals and flow around a cylinder. The mathematical foundations of the velocity potential and stream function are discussed in detail, giving readers a solid understanding of their applications in analyzing fluid motion.

Additionally, finite control volume analysis is introduced as an essential technique in fluid mechanics. The Reynolds Transport Theorem, conservation of mass, and Newton's second law are applied to control volumes, allowing for a detailed study of mass flow, momentum exchanges, and angular momentum in fluid systems. These methods are crucial for analyzing real-world engineering systems, including hydraulic systems, jet engines, and rotating machinery.

The final section of the document explores dimensional analysis and similitude, two fundamental tools in engineering that simplify complex systems into dimensionless forms. These techniques enable the development of generalized solutions and predictive models, particularly when experimental data is essential to validate theoretical findings.

Through these topics, this document builds on the principles introduced in "Fluid Mechanics 1", providing students with the advanced tools and methods necessary for solving more complex fluid dynamics problems in a variety of engineering contexts.

# FLUID KINEMATICS

## 1. INTRODUCTION

Fluid Kinematics focuses on the study of fluid motion, exploring the behavior of fluids without considering the forces that cause their movement. The chapter builds on the principles learned in Fluid Mechanics 1, delving deeper into key concepts such as fluid velocity, acceleration, and the various methods of flow description. Starting with the introduction of different reference systems like Cartesian, cylindrical, and spherical coordinates, it then covers critical tools for analyzing fluid motion, including the continuity equation, streamlines, pathlines, and streaklines. The chapter also discusses rotational and irrotational flows, introducing concepts such as vorticity, circulation, and potential flows. Additionally, students learn how to use the superposition principle to combine simple flows into more complex patterns and how graphical methods and complex potential theory can aid in analyzing real-world fluid dynamics problems.

By the end of the chapter, students should be able to:

- Understand the concept of fluid kinematics and its importance in fluid dynamics.
- Differentiate between Eulerian and Lagrangian fluid descriptions.
- Derive and apply the continuity equation for different types of flow.
- Identify and analyze streamlines, pathlines, and streaklines in steady and unsteady flows.
- Understand rotational and irrotational flows, vorticity, and circulation.
- Apply potential flow theory and superposition methods to analyze complex flow patterns.
- Use complex potential theory and conformal transformations in flow analysis.

## 2. REFERENCE SYSTEMS

In general, three independent spatial dimensions and time are needed to fully describe fluid motion. In three dimensions, Cartesian coordinates (Figure 1.1a) may be used to locate a point $P$ via the coordinate triplets $(x, y, z)$ with corresponding velocity components $(u, v, w)$. Cylindrical polar coordinates for $P$ (Figure 1.1b) are denoted by $(r, \theta, z)$ with corresponding velocity components $(v_r, v_\theta, v_z)$. ). Spherical polar coordinates for P (Figure 1.1c) are denoted by $(r, \theta, \varphi)$ with corresponding velocity components $(v_r, v_\theta, v_\varphi)$.

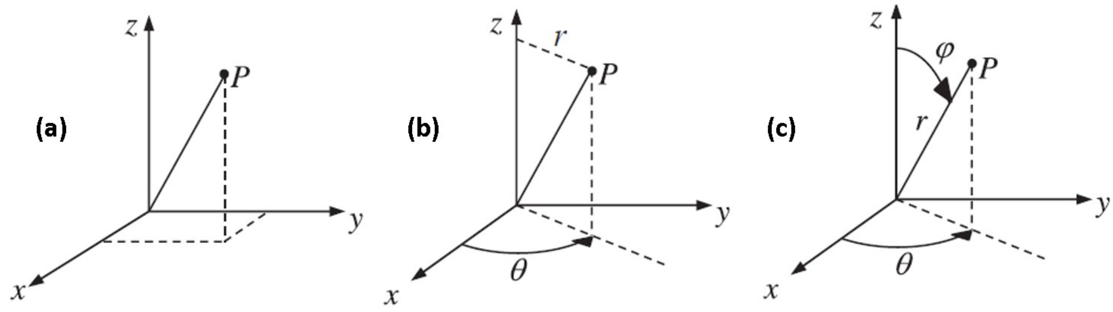



**Figure 1.1** Coordinate systems (a) Cartesian, (b) Cylindrical, (c) Spherical

- **Fluid Models**

Fluid models are categorized into two types: the Continuum model and the Molecular model. The continuum model treats fluids as a continuous medium, focusing on macroscopic properties like density and velocity without considering molecular interactions. In contrast, the molecular model examines fluid behavior at the molecular level, analyzing interactions between individual molecules. These models are used based on the scale of analysis and the specific fluid properties being studied. This course focuses on the continuum model where fluids are assumed to be made up of fluid particles interacting with each other and with their environment. A fluid particle is an infinitesimally small volume of fluid that is treated as a point for the purposes of analyzing its motion in the flow. It moves with the surrounding fluid, following the flow path, and its properties (like density, velocity, pressure, and temperature) change as it moves.

- **Flow Descriptions**

There are two distinct methods to describe fluid motion: the Eulerian and Lagrangian approaches. The Eulerian description focuses on observing fluid properties at fixed locations in space, such as measuring temperature or pressure at a stationary point over time. This method is practical for most theoretical and experimental analyses, providing insights into the flow behavior at specific points. As illustrated in Figure 1.1, one example of this method involves attaching a sensor at a fixed location, like the top of a chimney, to measure the temperature of the fluid as it passes by. On the other hand, the Lagrangian description follows individual fluid particles as they move through space, tracking their properties, such as temperature, along their path over time. For instance, Figure 1.2 also illustrates the Lagrangian method, showing how the temperature of a specific fluid particle (represented as particle A) is tracked as it moves, providing a dynamic view of the changes experienced by that particular particle over time. While the Eulerian method is more commonly used due to its simplicity and efficiency in various applications, the Lagrangian method is valuable when studying the motion and behavior of individual particles, such as in blood flow, or certain numerical simulations. These two descriptions, though distinct, are complementary, and data from one can often be used to derive information from the other. This course, however, primarily focuses on the Eulerian method, as it is the most widely applicable for the analysis of fluid flow in many practical situations.

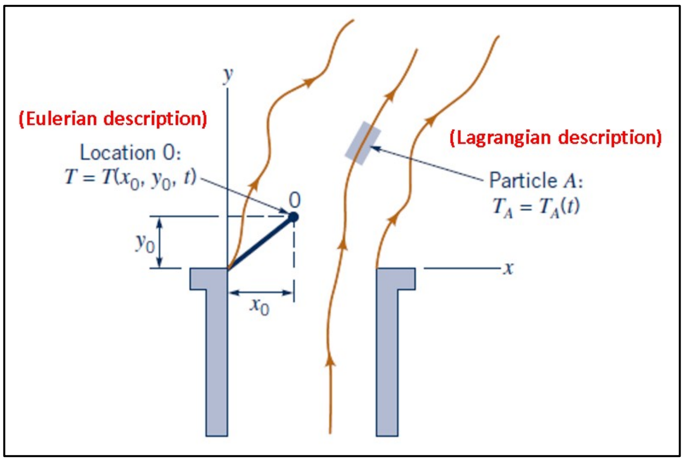



**Figure 1.2** Eulerian and Lagrangian descriptions of temperature of a flowing fluid

- **Velocity Field**

Consider an array of sensors that can simultaneously measure the magnitude and direction of fluid velocity at many fixed points within the flow as a function of time (Figure 1.3.a). In the limit of measuring velocity at all points within the flow, we would have sufficient information to define the velocity vector field:

$$\vec{V} = u(x, y, z, t)\vec{i} + v(x, y, z, t)\vec{j} + w(x, y, z, t)\vec{k}$$

Where $u$,$v$, and $w$ are the components of velocity in the $x$, $y$, and $z$ directions, respectively, and $\vec{\boldsymbol{\imath}}, \vec{\boldsymbol{\jmath}}, \vec{\boldsymbol{k}}$ are the unit vectors in these directions (see Figure 1.3.b). The magnitude of the velocity (speed) is given by:

$$V = \sqrt{u^2 + v^2 + w^2}$$

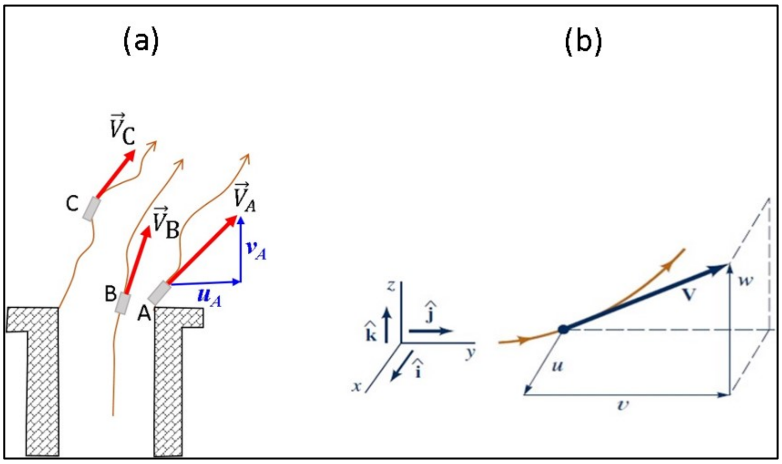



**Figure 1.3 (a)** velocity field at a given time $t$, **(b)** velocity vector representation

**Example 1.1**

A velocity field is given by $\vec{V} = (V_0 / \ell)(-x\vec{i} + y\vec{j})$ where $V_0$ and $\ell$ are constants.

At what location in the flow field is the speed equal to $V_0$ ?Make a sketch of the velocity field for $x \geq 0$ by drawing arrows representing the fluid velocity at representative locations.

**Solution**

The $x$, $y$, and $z$ components of the velocity are given by $u = -V_0 x / \ell$, $v = V_0 y / \ell$, $w = 0$ so that the fluid speed, $\boldsymbol{V}$, is:

$$V = \sqrt{u^2 + v^2 + w^2} = \frac{V_0}{\ell}\sqrt{x^2 + y^2} \qquad \text{(E.1.1)}$$

The speed is $V = V_0$ at any location on the circle of radius $\ell$ centered at the origin $\left[x^2 + y^2 = \ell^2\right]$ as shown in Figure. E1.1a.

The direction of the fluid velocity relative to the x axis is given in terms of $\theta = arctang\ (u/v)$ as shown in Figure E1.1b. For this flow:

$$\tan\theta = \frac{v}{u} = \frac{V_0 y / \ell}{-V_0 x / \ell} = -\frac{y}{x}$$

Thus, along the $x$ axis ($y = 0$) $tan\theta = 0$, so that $\theta = 0°$ or $\theta = 180°$. Similarly, along the $y$ axis ($x = 0$) we obtain $tan\theta = \pm\infty$, so that $\theta = 90°$ or $\theta = 270°$. Also, for $y = 0$ we find $V = (-V_0 x/\ell)\vec{i}$ , while for $x = 0$ we have $V = (V_0 y/\ell)\vec{j}$ , indicating (if $V_o > 0$) that the flow is directed away from the origin along the $y$ axis and toward the origin along the $x$ axis as shown in Figure 1.1a.

By determining $V$ and $\theta$ for other locations in the $x-y$ plane, the velocity field can be sketched as shown in the figure. For example, on the line $x = y$ the velocity is at a 45° angle relative to the $x$-axis ($tan\theta = v/u = -y/x = -1$). At the origin $x = y = 0$ so that this point is a stagnation point. The farther from the origin the fluid is, the faster it is flowing (as seen from Eq.E.1.1). By careful consideration of the velocity field, it is possible to determine considerable information about the flow.

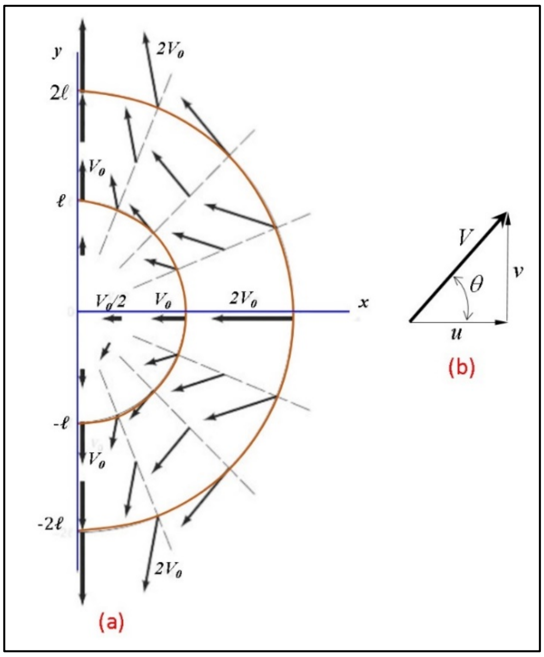



Figure E 1.1

- **Velocity Field Dimensions**

The velocity field can be one-dimensional (only $u$), two-dimensional ($u$ and $v$), or three-dimensional ($u$, $v$, and $w$) depending on the number of velocity components considered.

- **Steady vs. Unsteady Flows**

A steady flow occurs when the velocity field does not change with time, which is represented mathematically as $\frac{\partial \vec{V}}{\partial t} = 0$

An unsteady flow happens when the velocity changes with time, represented as $\frac{\partial \vec{V}}{\partial t} \neq 0$

- **Streamlines, Streaklines, and Pathlines**

**Streamlines** are lines that are everywhere tangent to the local velocity vector at a given instant (Figure 1.4a). In steady flow, they remain fixed in space, whereas in unsteady flow, they may change with time. Streamlines are important in analytical work, and they can be mathematically derived from the velocity field by integrating equations that define lines tangent to the velocity vectors. For 2-D flows, the streamline equation can be determined by integrating the slope equation (Figure 1.4b):

$$\left(\frac{dy}{dx}\right)_{\text{along a streamline}} = \frac{v}{u}$$

Where $u = u(x, y, z, t)$ and $v = v(x, y, z, t)$.

**A streamtube** is defined as a region within a flow field bounded by streamlines, where fluid particles move along these streamlines without crossing the boundaries (Figure 1.4c). Since streamlines are everywhere parallel to the local velocity, fluid cannot cross a streamline by definition.

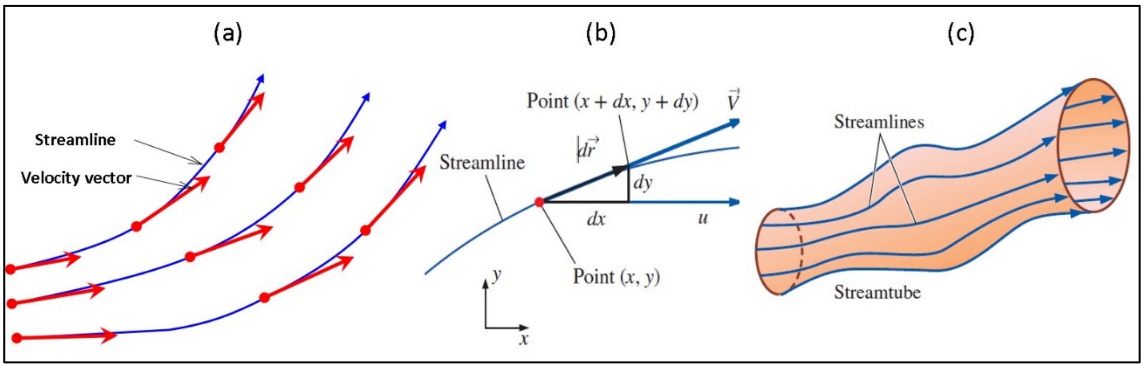



**Figure 1.4 (a)** streamlines, **(b)** plane streamline slop, **(c)** streamtube

**<u>Example 1.2</u>**

Consider the two-dimensional steady flow discussed in Example 4.1,

$$\vec{V} = (V_0 / \ell)(-x\vec{i} + y\vec{j})$$

Determine the streamlines for this flow.

**<u>Solution</u>**

Since $u=-V_0x/\ell$ and $v=V_0y/\ell$, it follows that streamlines are given by solution of the equation

$$\frac{dy}{dx}=\frac{v}{u}=\frac{V_0y/\ell}{-V_0x/\ell}=-\frac{y}{x}$$

in which variables can be separated and the equation integrated to give

$$\int\frac{dy}{y}=-\int\frac{dx}{x}$$

or

$$\ln y=-\ln x+constant$$

Thus, the streamlines equation is $y=\dfrac{C}{x}$, where $C$ is a constant.

By using different values of the constant $C$, we can plot various lines in the $x-y$ plan (the streamlines). The streamlines for $x\geq 0$ are plotted in Figure E 1.2.

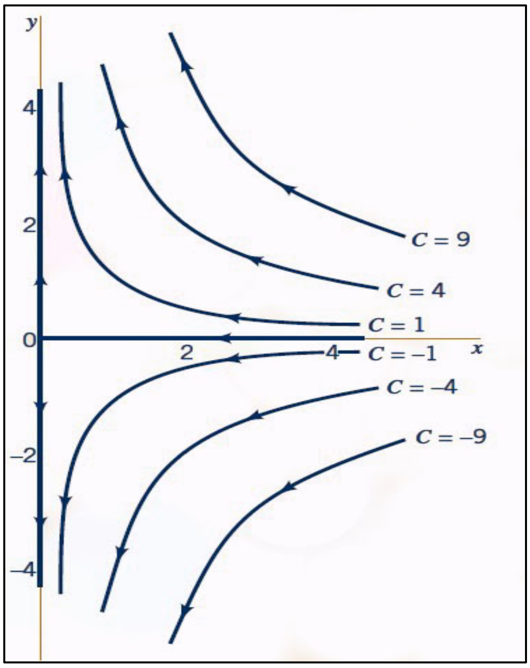



**Figure E 1.2**

**Streaklines** consist of all particles that have passed through a common point in the flow. They are typically used in experimental work, created by injecting a tracer (like dye or smoke) into the flow and observing the path of particles as illustrated in Figure 1.5. In steady flow, streaklines coincide with streamlines, but in unsteady flow, they may differ.

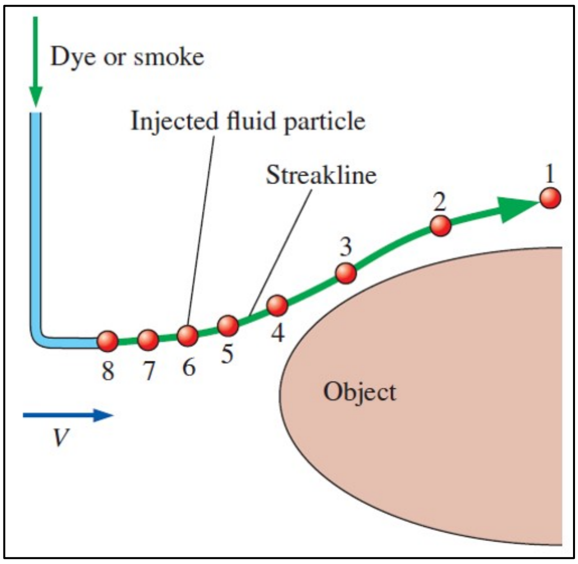



**Figure 1.5** Streakline illustration

**Pathlines** are the actual paths traced by individual fluid particles as they move through space (Figure 1.6). This Lagrangian concept can be observed in experiments by marking a fluid particle and tracking its movement over time. In steady flow, pathlines are identical to streamlines and streaklines, but in unsteady flows, they can differ.

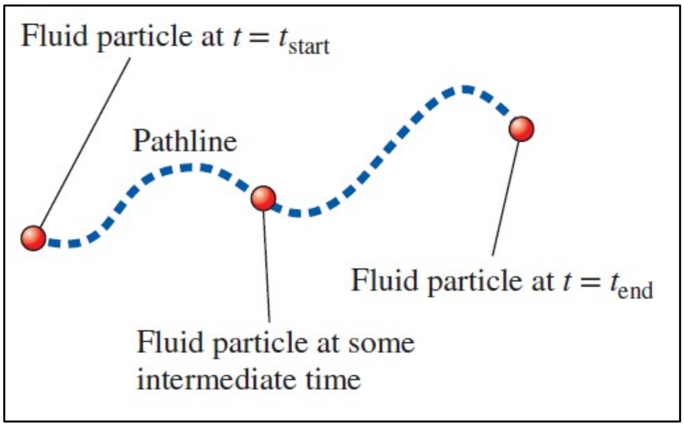



**Figure 1.6** Pathline illustration

<u>**Example 1.3**</u>

Water flowing from the oscillating slit shown in Figure E1.3a produces a velocity field given by $\vec{V} = u_0 \sin[\omega(t - y/v_0)]\vec{i} + v_0\vec{j}$, where $u_0$, $v_0$ and $\omega$ are constants. Thus,the $y$-component of velocity remains constant $(v = v_0)$ and the $x$-component of velocity at

$y = 0$ coincides with the velocity of the oscillating sprinkler head $[u = u_0 \sin(\omega t)$ at $y = 0]$.

Determine the streamline that passes through the origin at $t = 0$; at $t = \pi/2\,\omega$.

Determine the pathline of the particle that was at the origin at $t = 0$; at $t = \pi/2\,\omega$.

Discuss the shape of the streakline that passes through the origin

**Solution**

Since $u = u_0 \sin[\omega(t - y/v_0)]$ and $v = v_0$, it follows that streamlines are given by solution of the equation

$$\frac{dy}{dx} = \frac{v}{u} = \frac{v_0}{u_0 \sin[\omega(t - y/v_0)]}$$

or $(u_0 v_0/\omega)\cos[\omega(t - y/v_0)] = v_0 x + C$ (E.3.1)

Where $C$ is a constant. For the streamline at $t = 0$ that passes through the origin ($x = 0, y = 0$) the value of $C$ is obtained from Eq. E.3.1 as $C = u_0 v_0/\omega$. Hence, the equation for this streamline is:

$$x = \frac{u_0}{\omega}\left[\cos\left(\frac{\omega y}{v_0}\right) - 1\right] \quad \text{(E.3.2)}$$

Similarly, for the streamline at $t = \pi/2\,\omega$ that passes through the origin, Eq. E.3.1 gives $C = 0$. Thus, the equation for this streamline is:

$$x = \frac{u_0}{\omega}\left[\cos\omega\left(\frac{\pi}{2\omega} - \frac{y}{v_0}\right)\right] = \frac{u_0}{\omega}\left[\cos\left(\frac{\pi}{2} - \frac{\omega y}{v_0}\right)\right]$$

$$\Rightarrow \quad x = \frac{u_0}{\omega}\sin\left(\frac{\omega y}{v_0}\right) \quad \text{(E.3.3)}$$

These two streamlines, plotted in Figure E1.3b, are not the same because the flow is unsteady. At the origin ($x = 0, y = 0$) the velocity is $\vec{V} = u_0\,\vec{i}$ at $t = 0$ and $\vec{V} = u_0\,\vec{i} + v_0\,\vec{j}$ at $t = \pi/2\,\omega$. Thus, the angle of the streamline passing through the origin changes with time. Similarly, the shape of the entire streamline is a function of time.

The pathline of a particle can be obtained from the velocity field and the definition of the velocity. Since $u = dx/dt$, $v = dy/dt$, and we obtain:

$$\frac{dx}{dt} = u_0 \sin\left[\omega\left(t - \frac{y}{v_0}\right)\right] \text{ and } \frac{dy}{dt} = v_0$$

The $y$ equation can be integrated (since $v_0 = constant$) to give the $y$ coordinate of the pathline as:

$$y = v_0 t + C_1 \qquad \text{(E.3.4)}$$

where $C_1$ is a constant. With this known $y = y(t)$ dependence, the $x$ equation for the pathline becomes:

$$\frac{dx}{dt} = u_0 \sin\left[\omega\left(t - \frac{v_0 t + C_1}{v_0}\right)\right] = -u_0 \sin\left(\frac{C_1 \omega}{v_0}\right)$$

This can be integrated to give the $x$ component of the pathline as:

$$x = -\left[u_0 \sin\left(\frac{C_1 \omega}{v_0}\right)\right] t + C_2 \qquad \text{(E.3.5)}$$

Where $C_2$ is a constant. For the particle that was at the origin ($x = 0, y = 0$) at time $t = 0$, Eqs. E.2.4 and E.2.5 give $C_1 = C_2 = 0$. Thus, the pathline for this particle is:

$$x = 0 \text{ and } y = v_0 t \qquad \text{(E.3.6)}$$

Similarly, for the particle that was at the origin at $t = \pi/2\,\omega$, Eqs. E.3.4 and E.5.5 give $C_1 = -\pi v_0/2\,\omega$ and $C_2 = -\pi u_0/2\,\omega$. Thus, the pathline for this particle is:

$$x = u_0\left(t - \frac{\pi}{2\omega}\right) \text{ and } y = v_0\left(t - \frac{\pi}{2\omega}\right) \qquad \text{(E.3.7)}$$

The pathline can be drawn by plotting the locus of $x(t), y(t)$ values for $t \geq 0$ or by eliminating the parameter $t$ from Eq. E.3.7 to give:

$$y = \frac{v_0}{u_0} x \qquad \text{(E.3.8)}$$

The pathlines given by Eqs. E.3.6 and E.3.8, shown in Figure E4.3c, are straight lines from the origin (rays). The pathlines and streamlines do not coincide because the flow is unsteady.

The streakline through the origin at time $t = 0$ is the locus of particles at $t = 0$ that previously ($t < 0$) passed through the origin. The general shape of the streaklines can be seen as follows. Each particle that flows through the origin travels in a straight line (pathlines are rays from the origin), the slope of which lies between $\pm v_0/u_0$ as shown in Figure E4.3d. Particles passing through the origin at different times are located on different rays from the origin and at different distances from the origin. The net result is that a stream of dye continually injected at the origin (a streakline) would have the shape shown in Fig. E4.3d. Because of the unsteadiness, the streakline will vary with time, although it will always have the oscillating, sinuous character shown.

*In this example neither the streamlines, pathlines, nor streaklines coincide. If the flow were steady, all of these lines would be the same.*

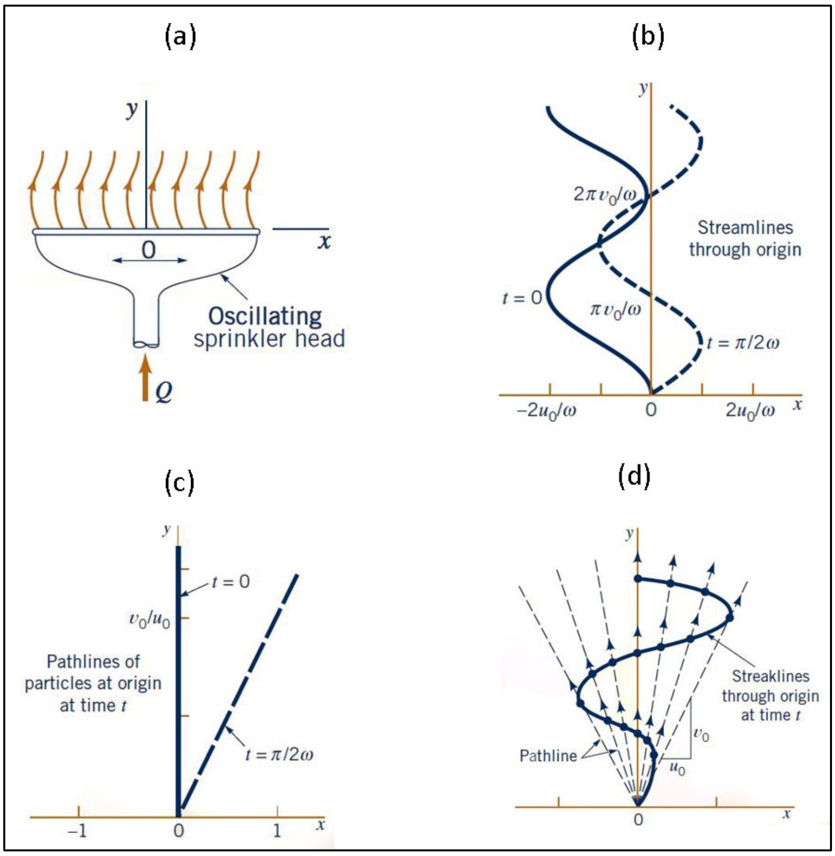



**Figure E.1.3**

## 3. CONTINUITY EQUATION-DIFFERENTIAL FORM

- **The Control Volume**

The control volume approach involves analyzing fluid flow by defining a spatially bounded region (control volume) through which fluid may pass, remain, or exit, with its boundary (control surface) separating it from the surroundings. This flexible region can be fixed, moving, or deforming, depending on the system being studied, and aligns with the Eulerian description, where fluid properties like velocity and pressure are observed at fixed spatial points rather than tracking individual particles. Widely used in fluid dynamics and machinery analysis (e.g., pumps, turbines), this approach enables practical evaluation of system performance by focusing on boundary fluxes and internal variations, bypassing the need to track individual fluid particles.

- **Continuity equation (differential form)**

Conservation of mass states that the net mass entering or leaving the control volume per unit time equals the rate of increase of mass within the control volume.

Consider the differential fixed control volume as shown in Figure 1.7. The mass flow rate in the $x$ −direction across the left side is $(\rho u)dydz$. The mass flow rate in the $x$ −direction across the right side, using a first-order approximation (since $dx$ is small), is $[\rho u + \frac{\partial}{\partial x}(\rho u)dx]dydz$. Figure 1.8 shows only the mass flows on the $x$ or left and right faces. The flows on the $y$ (bottom and top) and the $z$ (back and front) faces have been omitted to avoid cluttering up the drawing.

The net flux in the $x$ −direction over the differential volume is:

**Net mass flow rate in the x-direction** $=(\rho u)dydz - \left[\rho u + \frac{\partial}{\partial x}(\rho u)dx\right]dydz$

**Net mass flow rate in the x-direction** $= -\frac{\partial}{\partial x}(\rho u)dxdydz$

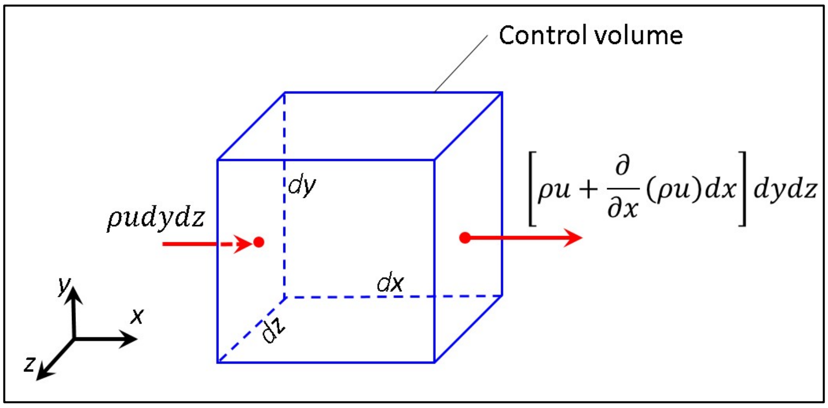



**Figure 1.7** Elemental fixed control volume showing the inlet and outlet mass flows on the $x$ faces.

Similarly, the net flux in the $y$ and $z$ −directions can be computed:

**Net mass flow rate in the y-direction** $= -\dfrac{\partial}{\partial y}(\rho v)dydxdz$

**Net mass flow rate in the x-direction** $= -\dfrac{\partial}{\partial z}(\rho w)dzdxdy$

Since the total rate of change of mass inside the control volume is equal to the net flow of mass into or out of the volume, we have the following equation:

$$\frac{\partial(\rho dxdydz)}{\partial t} = -\frac{\partial(\rho u)}{\partial x}dxdydz - \frac{\partial(\rho v)}{\partial y}dydxdz - \frac{\partial(\rho w)}{\partial z}dzdydx$$

After simplification, we obtain

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

or

$$\frac{\partial \rho}{\partial t} + \nabla(\rho\vec{V}) = 0$$

This equation, known as the continuity equation, expresses the principle of mass conservation in fluid flow.

**Cylindrical polar Coordinates**

The general continuity equation in cylindrical polar coordinates can be written as follows:

$$\frac{\partial \rho}{\partial t}+\frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r)+\frac{1}{r}\frac{\partial}{\partial \theta}(\rho v_\theta)+\frac{\partial}{\partial z}(\rho v_z)=0$$

**Special cases**

**Steady flow:** For steady flows $\frac{\partial}{\partial t} \equiv 0$ and all properties are functions of position only. Continuity equation reduces to

Cartesian: $\dfrac{\partial(\rho u)}{\partial x}+\dfrac{\partial(\rho v)}{\partial y}+\dfrac{\partial(\rho w)}{\partial z}=0$

Cylindrical: $\dfrac{1}{r}\dfrac{\partial}{\partial r}(r\rho v_r)+\dfrac{1}{r}\dfrac{\partial}{\partial \theta}(\rho v_\theta)+\dfrac{\partial}{\partial z}(\rho v_z)=0$

**Incompressible flow:** For incompressible flows, the density changes (in time and space) are negligible.

Cartesian: $\dfrac{\partial u}{\partial x}+\dfrac{\partial v}{\partial y}+\dfrac{\partial w}{\partial z}=0$

Cylindrical: $\dfrac{1}{r}\dfrac{\partial}{\partial r}(r v_r)+\dfrac{1}{r}\dfrac{\partial v_\theta}{\partial \theta}+\dfrac{\partial v_z}{\partial z}=0$

**Two-dimensional incompressible flow:**

Cartesian: $\dfrac{\partial u}{\partial x}+\dfrac{\partial v}{\partial y}=0$

Polar: $\dfrac{1}{r}\dfrac{\partial}{\partial r}(r v_r)+\dfrac{1}{r}\dfrac{\partial v_\theta}{\partial \theta}=0$

**Example 1.5**

The $x$ −component velocity is given by $u(x,y)=Ay^2$ in an incompressible plane flow. Determine $v(x,y)$ if $v(x,0)=0$.

**Solution**

The differential continuity equation for an incompressible, plane flow is

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

Since in a plane flow the two velocity components depend only on $x$ and $y$. Using the given $u(x, y)$ we find that

$$\frac{\partial(Ay^2)}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad \Rightarrow \quad \frac{\partial v}{\partial y} = 0$$

$$\Rightarrow \quad v(x, y) = f(x)$$

However, $v(x, 0) = 0$ requiring that $f(x) = 0$. Consequently,$v(x, y) = 0$

**Example 1.6**

The velocity components for a certain incompressible, steady-flow field are

$$\begin{cases} u = x^2 + y^2 + z^2 \\ v = xy + yz + z \\ w = ? \end{cases}$$

Determine the form of the $z$ component $w$ required to satisfy the continuity equation.

**Solution**

Any physically possible velocity distribution must satisfy conservation of mass as expressed by the continuity equation

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

$$\Rightarrow (2x) + (x + z) + \frac{\partial w}{\partial y} = 0$$

$$\Rightarrow \frac{\partial w}{\partial z} = -3x - z$$

$$\Rightarrow w = -3x - \frac{1}{2}z^2 + f(x, y)$$

The third velocity component cannot be explicitly determined since the function $f(x, y)$ can have any form and conservation of mass will still be satisfied. Some additional information is needed to completely determine $w$.

## 4. CONCEPTS OF VOLUMETRIC FLOW RATE AND MASS FLOW RATE

- **Continuity Equation based on Stream Tube Concept**

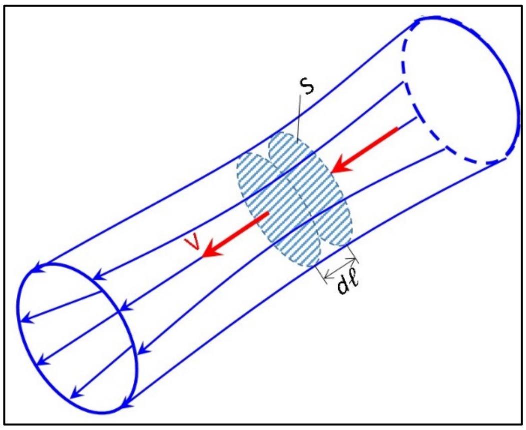
Consider the stream tube shown in Figure 1.8. The net mass flow through a small volume of cross-section $S$ and thickness $d\ell$ can be computed as follows:

Mass entering the volume: $\rho VS$

Mass leaving the volume: $\rho VS + \dfrac{\partial}{\partial \ell}(\rho VS)d\ell$

Where $V$ is the fluid velocity normal to cross-section $S$.

The mass change inside the volume during an infinitesimal time interval $dt$ is $\dfrac{\partial}{\partial t}(\rho S d\ell)$




.

**Figure 1.8** Flow through a stream tube showing the flow through an elemental volume of the tube

According to the conservation of mass principle, this variation is equal to the net flow of mass entering and leaving the volume:

$$\frac{\partial}{\partial t}(\rho S d\ell) = \rho VS - \left[\rho VS + \frac{\partial}{\partial \ell}(\rho VS)d\ell\right]$$

or

$$\frac{\partial}{\partial t}(\rho S) + \frac{\partial}{\partial \ell}(\rho VS) = 0$$

For steady flow $\frac{\partial}{\partial t} \equiv 0$, we have:

$$\frac{\partial}{\partial \ell}(\rho VS) = 0 \quad \Rightarrow \quad \rho VS = \textbf{constant}$$

That is, between two or more sections in the tube, we can write:

$$\rho_1 V_1 S_1 = \rho_2 V_2 S_2 = \rho_3 V_3 S_3 = \ldots = \dot{m}$$

**$\dot{m}$ is the fluid mass flow rate, which is constant along the stream tube.**

For incompressible flow $\rho = const.$,

$$V_1 S_1 = V_2 S_2 = V_3 S_3 = \ldots = q_v$$

**$q_v$ is the volume flow rate, which is constant along the stream tube.**

- **The Stream Function**

Consider the simple case of incompressible, two-dimensional flow in the $xy$ −plane. The continuity equation in Cartesian coordinates reduces to

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

A clever variable transformation enables us to rewrite this equation in terms of one dependent variable ($\psi$) instead of two dependent variables ($u$ and $v$). We define the stream function $\psi$ as:

$$u = \frac{\partial \psi}{\partial y} \quad \text{and} \quad v = -\frac{\partial \psi}{\partial x}$$

Substitution into the continuity equation yields

$$\frac{\partial}{\partial x}\left(\frac{\partial \psi}{\partial y}\right) + \frac{\partial}{\partial y}\left(-\frac{\partial \psi}{\partial x}\right) = \frac{\partial^2 \psi}{\partial x \partial y} - \frac{\partial^2 \psi}{\partial y \partial x} = 0$$

which is identically satisfied for any smooth function $\psi(x, y)$, because the order of differentiation ($y$ then $x$ versus $x$ then $y$) is irrelevant.

Since $\psi$ is a function of $x$ and $y$, its total differential can be written as follows:

$$d\psi = \frac{\partial \psi}{\partial x}dx + \frac{\partial \psi}{\partial y}dy = -vdx + udy$$

For a line along which the stream function is constant, we have $d\psi = 0$, which means that:

$$-vdx + udy = 0 \quad \Rightarrow \quad \left.\frac{dy}{dx}\right|_{\psi = const.} = \frac{v}{u}$$

This is exactly the streamlines equation. *Thus, the stream function $\psi$ is constant along streamlines as illustrated in Figure 1.9.*

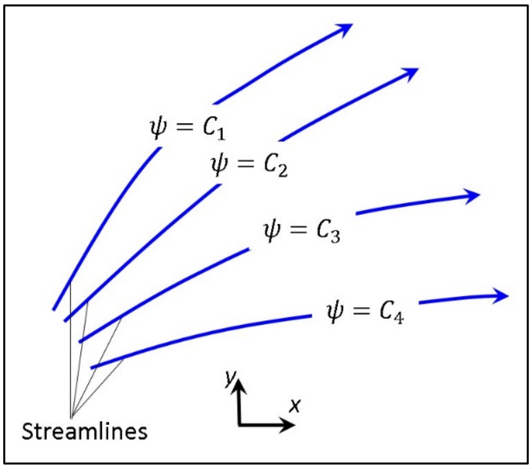



**Figure 1.9** Curves of constant stream function represent streamlines of the flow

There is another physically significant fact about the stream function: *The difference in the value of $\psi$ from one streamline to another is equal to the volume flow rate per unit width between the two streamlines.*

$$\Delta\psi = \psi_2 - \psi_1 = q_{v\,(1-2)}$$

This statement is illustrated in Figure 1.10. Consider two streamlines, $\psi_1$ and $\psi_2$ in a two-dimensional flow in the $xy$ −plane, of unit width into the page. By definition, no flow can cross a streamline. Thus, the fluid that happens to occupy the space between these two streamlines remains confined between the same two streamlines. It follows that the mass flow rate through any cross-sectional slice between the streamlines is the same at any instant in time (mass conservation). For steady, incompressible, two-dimensional flow in the $xy$ −plane, the volume flow rate $q_v$ between the two streamlines (per unit width) must therefore be a constant. If the two streamlines spread apart, as they do from cross-sectional slice $A$ to cross-sectional slice $B$, the average velocity between the two streamlines decreases accordingly, such that the volume flow rate remains the same, $q_v(A) = , q_v(B)$.

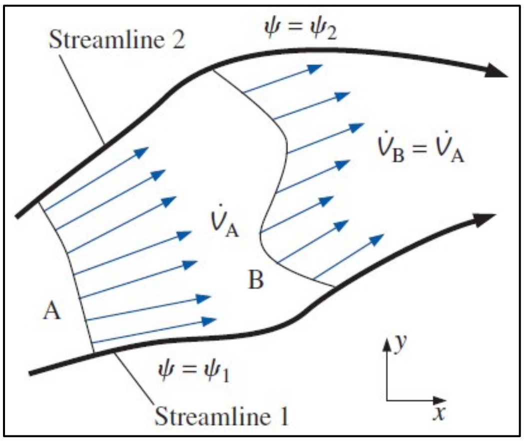



**Figure 1.10** Volume flow rate between two streamlines in the $xy$ −plane

Additionally, the stream function can be applied in cylindrical coordinates for problems involving radial and tangential velocity components. It is particularly useful in solving axisymmetric flows, where the flow has rotational symmetry about a central axis.

$(r, \theta)$ system coordinates: $v_r = \dfrac{\partial \psi}{r \partial \theta}$ and $v_\theta = -\dfrac{\partial \psi}{\partial r}$

$(r, z)$ system coordinates: $v_r = -\dfrac{1}{r}\dfrac{\partial \psi}{\partial z}$ and $v_z = \dfrac{1}{r}\dfrac{\partial \psi}{\partial r}$

Note that the stream function $\psi$ is defined such that it satisfies continuity equation for each case.

**Example 1.7**

Consider a steady, two-dimensional, incompressible velocity field with $u = ax + b$ and $v = -ay + cx$, where $a = 0.50\ s^{-1}$, $b = 1.5\ m/s, and\ c = 0.35\ s^{-1}$.

Generate an expression for the stream function and plot some streamlines of the flow in the upper-right quadrant.

**Solution**

The stream function is defined by the flowing expressions:

$$u = \frac{\partial \psi}{\partial y} \tag{E.7.1}$$

$$v = -\frac{\partial \psi}{\partial x} \tag{E.7.2}$$

We start by equation (E.7.1) or (E.7.2) (it doesn't matter which equation we choose, the solution will be identical). Starting, for example, with equation (E.7.1)

$$u = \frac{\partial \psi}{\partial y} = ax + b \tag{E.7.3}$$

Next we integrate with respect to $y$, noting that this is a partial integration, so we add an arbitrary function of the other variable, $x$, rather than a constant of integration,

$$\psi = axy + by + f(x)$$

Now we choose equation (E.7.2), differentiate equation (E.7.3), and rearrange as follows:

$$v = -\frac{\partial \psi}{\partial x} \Rightarrow -ay + cx = -ay - \frac{df(x)}{dx}$$

$$\Rightarrow \frac{df(x)}{dx} = -cx \Rightarrow f(x) = -\frac{c}{2}x^2 + C$$

$$\Rightarrow \psi = axy + by - \frac{c}{2}x^2 + C \tag{E.7.4}$$

To plot the streamlines, we note that equation (E.7.4) represents a family of curves, one unique curve for each value of $\psi$. Since $C$ is arbitrary, it is common to set it equal to zero, although it can be set to any desired value. For simplicity we set $C = 0$ and solve equation (E.7.4) for $y$ as a function of $x$, yielding

$$y = \frac{cx^2/2 + \psi}{(ax + b)} \tag{E.7.5}$$

Which represents the streamlines equation.

For the given values of constants $a$, $b$, and $c$, we plot equation (E.7.5) for several values of $\psi$ in Figure 7.1.

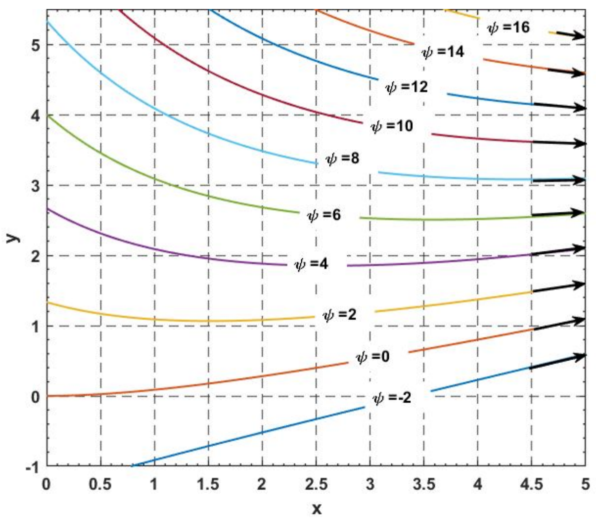



**Figure E.1.7**

## 5. ROTATIONAL AND IRROTATIONAL FLOWS

### 5.1 Vorticity

A closely related kinematic property of great importance to the analysis of fluid flows is the vorticity vector, defined mathematically as the $curl$ of the velocity vector $\vec{\boldsymbol{V}}$,

$$\Omega = \nabla \times \vec{V} = curl(\vec{V})$$

In Cartesian coordinates, $curl\ (\vec{V})$ is expanded as follows:

$$curl(\vec{V}) = \begin{vmatrix} \vec{i} & \vec{j} & \vec{k} \\ \dfrac{\partial}{\partial x} & \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial z} \\ u & v & w \end{vmatrix} = \left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)\vec{i} + \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)\vec{j} + \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\vec{k}$$

Physically, you can tell the direction of the vorticity vector by using the right-hand rule for cross product (Figure 1.11).

If the flow is two-dimensional in the $xy$-plane, the $z$-component of velocity $(w)$ is zero and neither $u$ nor $v$ varies with $z$. Thus, the first two components are identically zero and the vorticity reduces to

$$\Omega = \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\vec{k}$$

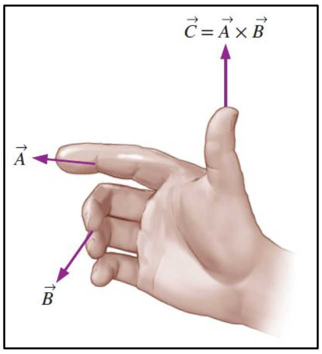



**Figure 1.11** The direction of a vector cross product

- **Relation to Fluid Particle Rotation**

Fluid particle rotation refers to the tendency of individual fluid particles to rotate around their center of mass as the fluid flows.

Vorticity quantifies this rotation. In regions of a flow where the vorticity is zero, the flow is said to be irrotational. Otherwise, the flow is said to be rotational. This means that the fluid particles do not just translate (move in a straight line), but they also rotate about an axis.

The vorticity vector points along the axis of rotation of the fluid, and its magnitude gives the strength of the rotation. In simple terms, if the fluid has high vorticity, the fluid particles are rotating more strongly.

**Example 1.8**

Consider the following steady, incompressible, two-dimensional velocity field:

$$\vec{V} = (u, v) = x^2\vec{i} + (-2xy - 1)\vec{j}$$

Is this flow rotational or irrotational?

**Solution**

The vorticity $\Omega$ for this flow is:

$$\Omega = \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\vec{k} = -2y - 0 = -2y$$

Since the vorticity is nonzero, this flow is rotational.

**Cylindrical coordinates**

In cylindrical coordinates, $(r, \theta, z)$, the vorticity vector is given by:

$$\Omega = \left(\frac{1}{r}\frac{\partial v_z}{\partial \theta} - \frac{\partial v_\theta}{\partial z}\right)\vec{e}_r + \left(\frac{\partial v_r}{\partial z} - \frac{\partial v_z}{\partial r}\right)\vec{e}_\theta + \frac{1}{r}\left(\frac{\partial (r v_\theta)}{\partial r} - \frac{\partial v_r}{\partial \theta}\right)\vec{e}_z$$

For two-dimensional flow in the $r\theta$-plane, the vorticity reduces to

$$\Omega = \frac{1}{r}\left(\frac{\partial (r v_\theta)}{\partial r} - \frac{\partial v_r}{\partial \theta}\right)\vec{e}_z$$

**Example 1.9**

A simple two-dimensional velocity field called a **line sink** is often used to simulate fluid being sucked into a line along the $z$ −axis. Suppose the volume flow rate per unit length along the $z$ −axis, $q_v/l$, is known, where $q_v$ is a negative quantity. In two dimensions in the r$\theta$-plane,

$$v_r = \frac{q_v}{2\pi l}\frac{1}{r} \quad \text{and} \quad v_\theta = 0$$

Draw several streamlines of the flow and calculate the vorticity. Is this flow rotational or irrotational?

**Solution**

Since there is only radial flow and no tangential flow, we know immediately that all streamlines must be rays into the origin. Several streamlines are sketched in Figure E.1.9.

The vorticity is:

$$\Omega = \frac{1}{r}\left(\frac{\partial (r v_\theta)}{\partial r} - \frac{\partial v_r}{\partial \theta}\right)\vec{e}_z = \frac{1}{r}(0-0)\vec{e}_z = 0$$

Since the vorticity vector is everywhere zero, this flow field is **irrotational**.

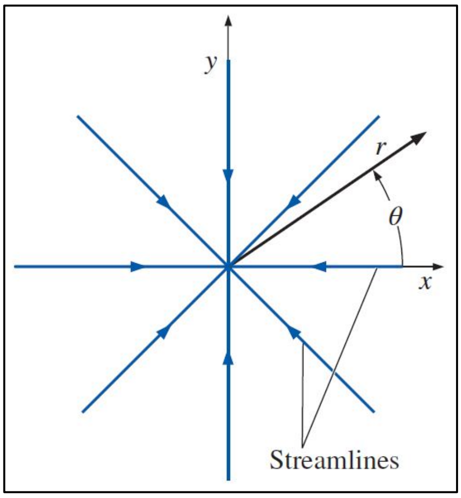



**Figure E.1.9**

## 5.2 Circulation

With reference to Figure 1.12, the circulation is defined as the counterclockwise line integral, around a closed curve $C$, of arc length $ds$ times the velocity component tangent to the curve.

$$\Gamma = \oint \vec{V}.ds$$

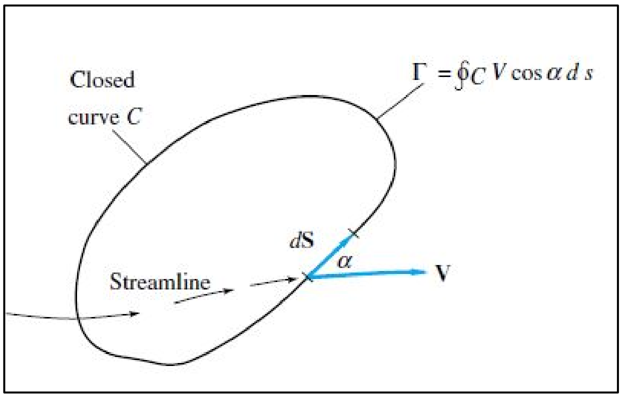



**Figure 1.12** Definition of the fluid circulation $\Gamma$

Circulation gives a measure of the intensity of rotational motion in the fluid. A high circulation value indicates strong rotation (like in a vortex), while a zero circulation indicates non-rotational flow (like in a uniform, straight flow).

If the circulation around the curve is nonzero, it indicates that the flow has a rotational component. This means that the fluid particles are rotating around the curve, and there is vorticity present in the flow.

In the case of zero circulation, the flow is said to be irrotational, meaning there is no net rotation of the fluid particles.

Circulation is directly related to the vorticity of the flow Ω. In a 2D flow, vorticity is related to the circulation around an infinitesimal closed loop by the Stokes' theorem:

$$\Gamma = \oint \vec{V}.ds = \iint_A \Omega.dA$$

Where $A$ is the surface bounded by the closed curve $C$, and $dA$ is the area element. This shows how circulation can be used to compute the vorticity over a surface.

## 6. POTENTIAL FLOWS (IRROTATIONAL FLOWS)

In the previous section, we defined irrotational flow as flow for which vorticity is zero,

$$\Omega = \nabla \times \vec{V} = curl(\vec{V}) = 0 \qquad (1.1)$$

On the other hand, we know from mathematics that the following identity is always valid for any scalar function $\phi(x, y)$:

$$curl(\nabla\phi) = 0 \quad (1.2)$$

By comparing equations (2.1) and (2.2), we deduce that the velocity field in an irrotational flow can be written as:

$$\vec{V} = \nabla\phi = \frac{\partial\phi}{\partial x}\vec{i} + \frac{\partial\phi}{\partial y}\vec{j} \qquad (1.3)$$

or

$$u = \frac{\partial\phi}{\partial x} \text{ and } v = \frac{\partial\phi}{\partial y} \quad (1.4)$$

And in cylindrical polar coordinates,

$$v_r = \frac{\partial\phi}{\partial r} \text{ and } v_\theta = \frac{\partial\phi}{r\partial\theta} \qquad (1.5)$$

The function $\phi(x, y)$ is called the ***velocity potential***, and the flow is said to be potential. Thus, for any irrotational flow, there is a potential function $\phi(x, y)$ such that the velocity field in the flow is derived from this potential, as indicated by Eqs. 2.4 or 2.5.

The continuity equation, for an incompressible flow, is given by:

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

Substituting $u$ and $v$ by their expressions given by Eq. 2.4, we obtain:

$$\frac{\partial^2 \phi}{\partial x^2}+\frac{\partial^2 \phi}{\partial y^2}=\nabla^2\phi=0 \quad (1.6)$$

and in cylindrical polar coordinates:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial \phi}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 \phi}{\partial r^2}=0 \qquad (1.7)$$

Eqs. 2.6 or 2.7 are called Laplace's equations.

Hence, the potential function satisfies Laplace's equation, and conversely, if a function $\phi(x,y)$ satisfies Laplace's equation, then it can represent a potential flow.

Note that Laplace's equation is linear, i.e. if two or more functions satisfy Laplace's equation, then the sum of these functions also satisfies Laplace's equation. In other words, the superposition of a number of potential flows is a potential flow. We have:

$$\nabla^2(\phi_1+\phi_2+\phi_3+\ldots)=\nabla^2\phi_1+\nabla^2\phi_2+\nabla^2\phi_3+\ldots=0 \quad (1.8)$$

## 7. PLANAR FLOWS

- **Potential And Stream Functions**

For 2D potential flows, the stream function $\psi(x,y)$ is defined by

$$u=\frac{\partial \psi}{\partial y} \text{ and } v=-\frac{\partial \psi}{\partial x} \qquad (1.9)$$

The continuity equation is automatically satisfied, and irrotationality leads to the Laplace equation,

$$\Omega=\frac{\partial u}{\partial y}-\frac{\partial v}{\partial x}=0$$

$$\Rightarrow \quad \frac{\partial^2 \psi}{\partial x^2}+\frac{\partial^2 \psi}{\partial y^2}=0 \quad (1.10)$$

*The similarity between Eqs. 2.6 and 2.10 implies that the velocity potential $\phi$ and the stream function $\psi$ of a given flow can describe another potential flow by interchanging $\phi$ and $\psi$.*

From Eqs. 2.4 and 2.9 we obtain the well-known Cauchy-Riemann equations or conditions:

$$\frac{\partial \phi}{\partial x} = \frac{\partial \psi}{\partial y} \text{ and } \frac{\partial \phi}{\partial y} = -\frac{\partial \psi}{\partial x} \quad (1.11)$$

- **Potential lines and streamlines**

The streamlines equation is defined by:

$$d\psi = 0 \Rightarrow -vdx + udy = 0$$

$$\Rightarrow \left(\frac{dy}{dx}\right)_{\psi} = \frac{v}{u} \; (1.12)$$

The potential lines equation is defined by:

$$d\phi = 0 \Rightarrow udx + vdy = 0$$

$$\Rightarrow \left(\frac{dy}{dx}\right)_{\phi} = -\frac{u}{v} \qquad (1.13)$$

By comparing Eqs. 2.12 and 2.13, the following can be written:

$$\left(\frac{dy}{dx}\right)_{\psi} = -\frac{1}{\left(\frac{dy}{dx}\right)_{\phi}} \quad (1.14)$$

Eq. 2.14 shows that the tangents to the streamlines and equipotential lines at the intersection points are perpendicular as shown in Figure 1.13 (Recall that two lines are orthogonal if the product of their slopes is -1). They are said to be orthogonal.

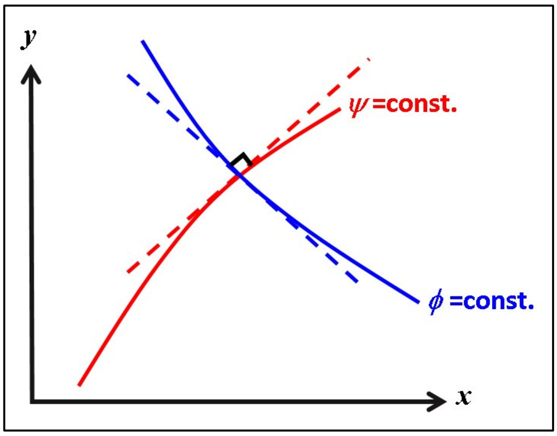



**Figure 1.13** Equipotential lines are orthogonal to streamlines.

**Example 2.1**

The velocity field of a plane incompressible flow is given by:

$u = 4x$ and $v = -4y$

Check that the continuity equation is satisfied

Show that the flow is irrotational

Determine the stream function and velocity potential

Plot the flow net in the upper-right quadrant ($x > 0$ and $y > 0$)

**Solution**

The continuity equation for two-dimensional incompressible flow is:

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

Substituting $u$ and $v$ with their expressions yields:

$$\frac{\partial(4x)}{\partial x} + \frac{\partial(-4y)}{\partial y} = 4 + (-4) = 0$$

The continuity equation is then satisfied

The vorticity in two-dimensional flow is defined as:

$$\Omega = \frac{\partial u}{\partial y} - \frac{\partial v}{\partial x}$$

Substituting $u$ and $v$ with their expressions yields:

$$\Omega = \frac{\partial(4x)}{\partial y} - \frac{\partial(-4y)}{\partial x} = 0 + 0 = 0$$

Since $\Omega = 0$, the flow is **irrotational**.

The stream function $\psi$ is related to the velocity components as:

$$u = \frac{\partial \psi}{\partial y} \text{ and } v = -\frac{\partial \psi}{\partial x}$$

From $u = 4x$, we integrate with respect to $\boldsymbol{y}$:

$$\frac{\partial \psi}{\partial y} = 4x \quad \Rightarrow \quad \psi = 4xy + f(x)$$

Now, from $v = -4y$ we differentiate $\psi = 4xy + f(x)$ with respect to $x$

$$\frac{\partial \psi}{\partial x} = 4y + \frac{df(x)}{dx} = 4y$$

$$\Rightarrow f(x) = C$$

Where $C$ is an arbitrary constant. It is common to set it equal to zero, although it can be set to any desired value. For simplicity we set $C = 0$.

$$\Rightarrow \quad \psi = 4xy$$

The velocity potential can be determined from its definition as a function of $u$ and $v$ components, or from the Cauchy-Riemann equations.

$$\frac{\partial \phi}{\partial x} = \frac{\partial \psi}{\partial y} = 4x \text{ and } \frac{\partial \phi}{\partial y} = -\frac{\partial \psi}{\partial x} = -4y$$

From $\partial\phi/\partial x = 4x$, we integrate with respect to $\boldsymbol{x}$:

$$\phi = 2x^2 + f(y)$$

Now, from $\partial\phi/\partial y = -4y$ we differentiate $\phi = 2x^2 + f(y)$ with respect to $y$

$$\frac{\partial \phi}{\partial y} = \frac{df(y)}{dy} = -4y$$

$\Rightarrow f(y) = -2y^2 + C$

Where $C$ is an arbitrary constant. For simplicity we set $C = 0$.

$\Rightarrow \phi = 2x^2 + -2y^2$

We will now plot the streamlines and equipotential lines based on the equations:

Streamlines : $\psi = const. \;\; \Rightarrow \;\; y = \dfrac{\psi}{4x}, \; \psi = C_1, \; C_2, \; C_3, ...$

Equipotential lines: $\phi = const. \Rightarrow y^2 = x^2 - \phi/2, \; \phi = C_1, \; C_2, \; C_3, ...$

The flow net, in the upper-right quadrant ($x > 0$ and $y > 0$), is shown in Figure E.2.1

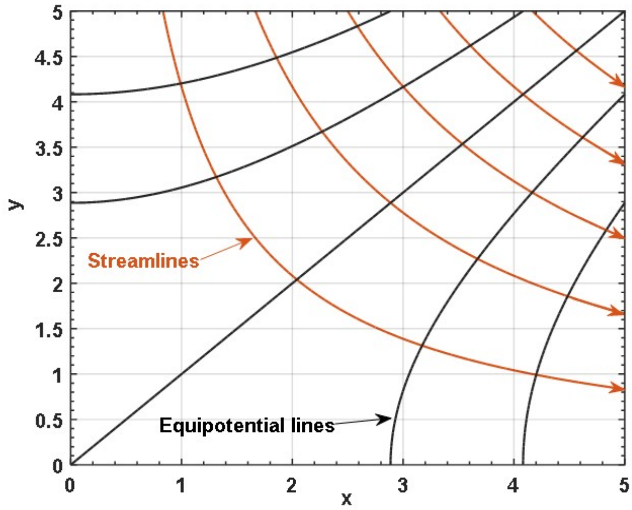



**Figure E.2.1**

## 8. ELEMENTARY POTENTIAL FLOWS

- **Uniform Flow**

The simplest plane flow is one for which the streamlines are all straight and parallel, and the magnitude of the velocity is constant $\vec{V} = (U, 0)$. This type of flow is called a uniform flow. For this flow, we have:

$$\frac{\partial \phi}{\partial x} = U = \frac{\partial \psi}{\partial y} \text{ and } \frac{\partial \phi}{\partial y} = 0 = -\frac{\partial \psi}{\partial x}$$

Therefore,

$$\phi = U\,x \qquad (1.15)$$

and

$$\psi = U\,y \qquad (1.16)$$

Where the arbitrary integration constants are taken to be zero at the origin.

Figure 1.14 presents a uniform flow in the $x$ −direction and in an arbitrary direction $\alpha$.

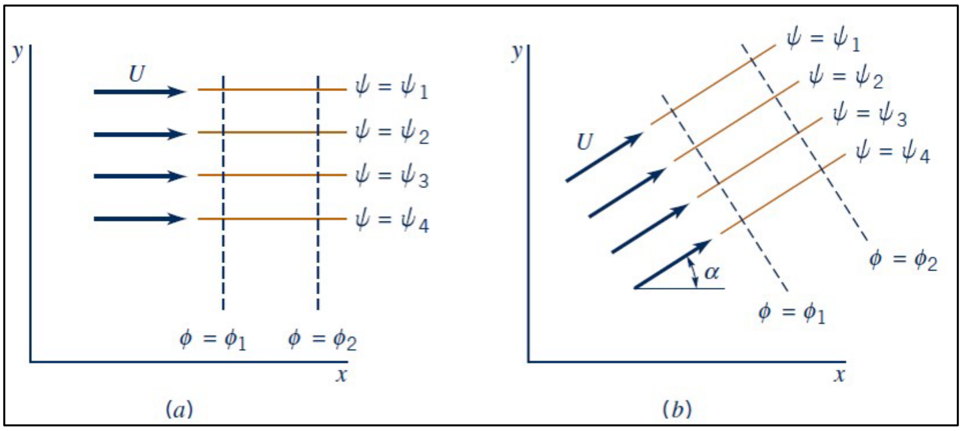



**Figure 1.14** Uniform flow **(a)** in the $x$ −direction, **(b)** in an arbitrary direction $\alpha$.

- **Source (Sink)**

Consider a fluid flowing radially outward from a line through the origin perpendicular to the $x-y$ plane as is shown in Figure 1.15. Let $q_v$ be the volume rate of flow emanating from the line (per unit length), and therefore to satisfy conservation of mass

$2\pi r.v_r = q_v$ (source if $q_v$ is positive and sink if negative)

Therefore, $v_r = \dfrac{q_v}{2\pi r}$ , $v_\theta = 0$

$$\frac{\partial\phi}{\partial r} = v_r = \frac{q_v}{2\pi r} = \frac{\partial\psi}{r\partial\theta} \quad \text{and} \quad \frac{\partial\phi}{r\partial\theta} = v_\theta = 0 = -\frac{\partial\psi}{\partial r}$$

The integration leads to

$$\phi = \frac{q_v}{2\pi}\ln r \qquad (1.17)$$

and

$$\psi = \frac{q_v}{2\pi}\theta \qquad (1.18)$$

It is apparent from Eq. 2.17 that the streamlines are radial lines, and from Eq.2.18 that the equipotential lines are concentric circles centered at the origin.

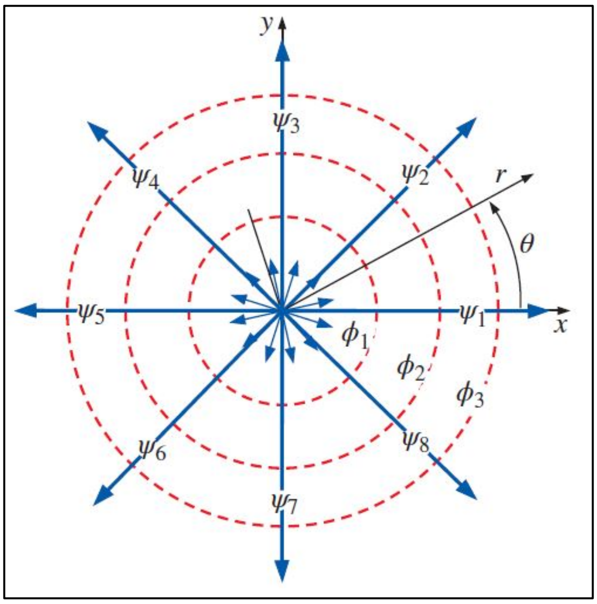



**Figure 1.15** The flow net for a source

- **Free Vortex**

A Free Vortex is a type of vortex in which the fluid particles rotate around a central point (see Figure 1.16), with the tangential velocity depending inversely on the radial distance from the center.

The velocity profile of a free vortex is given by:

$$v_r = 0 \text{ and } v_\theta = \frac{\Gamma}{2\pi r} \quad (1.19)$$

where $\Gamma$ is the circulation around the vortex, which is a constant.

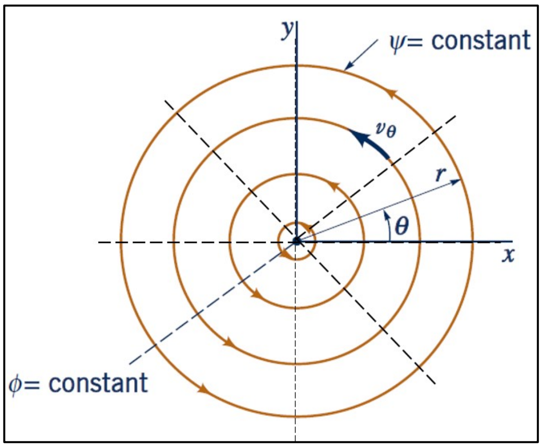



**Figure 1.16** The flow net for a source

The potential and stream function are determined as follows:

$$\frac{\partial \phi}{\partial r} = v_r = 0 = \frac{\partial \psi}{r\partial \theta} \quad \text{and} \quad \frac{\partial \phi}{r\partial \theta} = v_\theta = \frac{\Gamma}{2\pi r} = -\frac{\partial \psi}{\partial r}$$

The integration leads to:

$$\phi = \frac{\Gamma}{2\pi}\theta \qquad (1.20)$$

and

$$\psi = -\frac{\Gamma}{2\pi}\ln r \qquad (1.21)$$

## 9. SUPERPOSITION OF SIMPLE POTENTIAL FLOWS

Because the potential and stream functions satisfy the linear Laplace equation, the superposition of two potential flow is also a potential flow. From this, it is possible to construct potential flows of more complex geometry.

- **Source and sink**

Consider a source $q_v$ at $(-a, 0)$ combined whith a sink of equal strength $-q_v$ placed at $(a, 0)$ as shown in Figure 1.17.

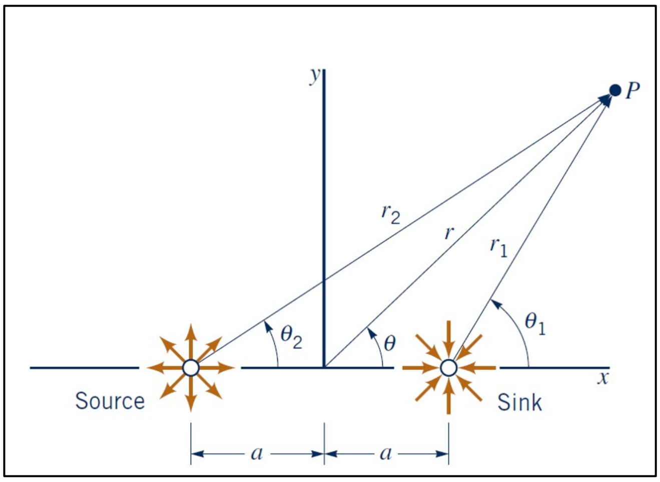



**Figure 1.17** Superposition of a source and a sink of equal strengths

The composite stream and potential functions are expressed as follows:

$$\psi = \psi_{source} + \psi_{sink} \quad \text{and} \quad \phi = \phi_{source} + \phi_{sink}$$

Using equations 2.17 and 2.18 and with reference to the polar coordinate system shown in figure 2.5, we write:

$$\psi = \frac{q_v}{2\pi}\theta_2 - \frac{q_v}{2\pi}\theta_1 \text{ and } \phi = \frac{q_v}{2\pi}\ln r_2 - \frac{q_v}{2\pi}\ln r_1$$

$$\Rightarrow \quad \psi = \frac{q_v}{2\pi}(\theta_2 - \theta_1) \text{ and } \phi = \frac{q_v}{2\pi}\ln(r_2/r_1)$$

We have, $tan\,\theta_2 = \dfrac{r\,sin\,\theta}{r\,cos\,\theta + a}$ ; $tan\,\theta_1 = \dfrac{r\,sin\,\theta}{r\,cos\,\theta - a}$

and, $tan(\theta_2 - \theta_1) = \dfrac{tan\,\theta_2 - tan\,\theta_1}{1 + tan\,\theta_2\,tan\,\theta_1}$

Therefore, $tan(\theta_2 - \theta_1) = \dfrac{-2a\,r\,sin\,\theta}{r^2 - a^2}$

or, $(\theta_2 - \theta_1) = tan^{-1}\left(\dfrac{-2a\,r\,sin\,\theta}{r^2 - a^2}\right) = -tan^{-1}\left(\dfrac{2a\,r\,sin\,\theta}{r^2 - a^2}\right)$

Hence,

$$\psi = -\frac{q_v}{2\pi}tan^{-1}\left(\frac{2a\,r\,sin\,\theta}{r^2 - a^2}\right) \quad (1.22)$$

or, in Cartesian coordinates,

$$\psi = -\frac{q_v}{2\pi}tan^{-1}\left(\frac{2a\,y}{x^2 + y^2 - a^2}\right) \quad (1.23)$$

Next, using the geometric relations,

$$r_2^2 = (r\sin\theta)^2 + (r\cos\theta + a)^2 = r^2 + a^2 + 2ar\cos\theta$$
$$r_1^2 = (r\sin\theta)^2 + (r\cos\theta - a)^2 = r^2 + a^2 - 2ar\cos\theta$$

to express the velocity potential as follows:

$$\phi = \frac{q_v}{2\pi}ln\left(\frac{\sqrt{r^2 + a^2 + 2ar\,cos\,\theta}}{\sqrt{r^2 + a^2 - 2ar\,cos\,\theta}}\right)$$

$$\phi = \frac{1}{2}\left(\frac{q_v}{2\pi}\right)ln\left(\frac{r^2 + a^2 + 2ar\,cos\,\theta}{r^2 + a^2 - 2ar\,cos\,\theta}\right) \quad (1.24)$$

or, in Cartesian coordinates,

$$\phi = \frac{1}{2}\left(\frac{q_v}{2\pi}\right) ln\left(\frac{(x+a)^2 + y^2}{(x-a)^2 + y^2}\right) \qquad (1.25)$$

The velocity components, using Eqs. 2.4 or 2.9, are:

$$u = \frac{q_v}{2\pi}\left(\frac{x+a}{(x+a)^2 + y^2} - \frac{x-a}{(x-a)^2 + y^2}\right) \qquad (1.26)$$

$$v = \frac{q_v}{2\pi}\left(\frac{y}{x^2 + (y+a)^2} - \frac{y}{x^2 + (y-a)^2}\right) \qquad (1.27)$$

The flow net for this flow is illustrated in Figure 1.18.

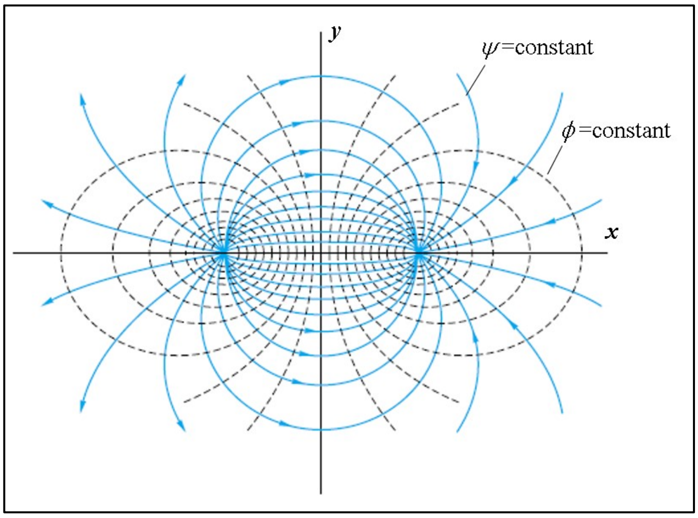



**Figure 1.18** The flow net for a source and sink combination

- **Doublet**

The so-called doublet is formed by letting the source and sink approach one another ($a \to 0$) while increasing the strength $q_v$ ($q_v \to \infty$) so that the product $q_v a$ remains constant. In this case,

$$tan^{-1}\left(\frac{2a\,r\,sin\,\theta}{r^2 - a^2}\right) \simeq \frac{2a\,r\,sin\,\theta}{r^2 - a^2}$$

$$\Rightarrow \psi \simeq -\frac{q_v}{2\pi}\frac{2a\,r\,sin\,\theta}{r^2 - a^2} = \frac{q_v a}{\pi}\frac{r\,sin\,\theta}{r^2 - a^2}$$

Since ($a \to 0$),

$$\psi = -K \frac{\sin\theta}{r} \quad (1.28)$$

Where $K$, a constant equal to $q_v a/\pi$ is called the strength of the doublet. The corresponding velocity potential for the doublet is:

$$\phi = K \frac{\cos\theta}{r} \quad (1.29)$$

The corresponding velocity components are:

$$v_r = -K \frac{\cos\theta}{r^2} \quad (1.30)$$

$$v_\theta = -K \frac{\sin\theta}{r^2} \quad (1.31)$$

A plot of the streamlines and equipotential lines is shown in Figure 1.19.

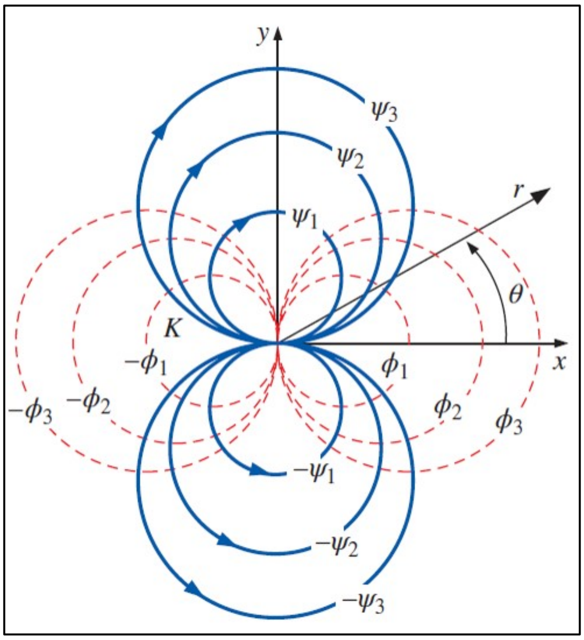



**Figure 1.19** Flow net for a doublet of strength $K$ located at the origin

- **Source in a Uniform Stream—Half-Body**

Assuming the uniform flow $U$ is in $x$ −direction and the source of $q_v$ at the origin as shown in Figure 1.20.

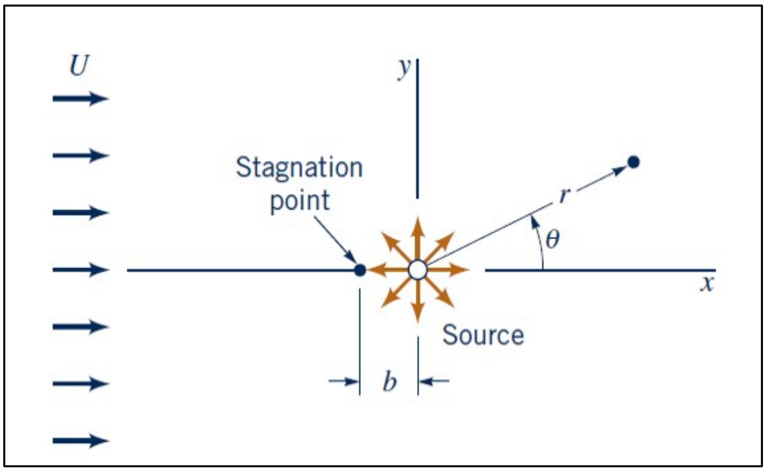



**Figure 1.20** superposition of a source and a uniform flow

The corresponding stream function and the velocity potential are:

$$\psi = Uy + \frac{q_v}{2\pi}\theta = Ur\sin\theta + \frac{q_v}{2\pi}\theta \qquad (1.32)$$

$$\phi = Ux + \frac{q_v}{2\pi}\ln r = Ur\cos\theta + \frac{q_v}{2\pi}\ln r \quad (1.33)$$

The velocity components are:

$$v_r = U\cos\theta + \frac{q_v}{2\pi r} \qquad (1.34)$$

$$v_\theta = U\sin\theta \qquad (1.35)$$

It is clear that a stagnation point ($v_r = v_\theta = 0$) occurs at $\theta = \pi$ and $r = b = q_v/(2\pi U)$.

Therefore, the streamline passing through the stagnation point is: $\psi_s = \frac{q_v}{2} = \pi bU$

It follows that the equation of the streamline passing through the stagnation point is

$$\pi bU = Ur\sin\theta + \frac{q_v}{2\pi}\theta$$

or

$$r = \frac{b(\pi - \theta)}{\sin\theta}$$

where $\theta$ can vary between 0 and $2\pi$.

The maximum height of the curve $\psi_s$ is :

$$h = r\sin\theta = \frac{q_v}{2U} = \pi b \quad as \ \ \theta \to 0 \text{ and } \ r \to \infty$$

A plot of this streamline is shown in Figure 1.21. If we replace this streamline with a solid boundary, as indicated in the figure, then it is clear that this combination of a uniform flow and a source can be used to describe the flow around a streamlined body placed in a uniform stream. The body is open at the downstream end and thus is called a half-body.

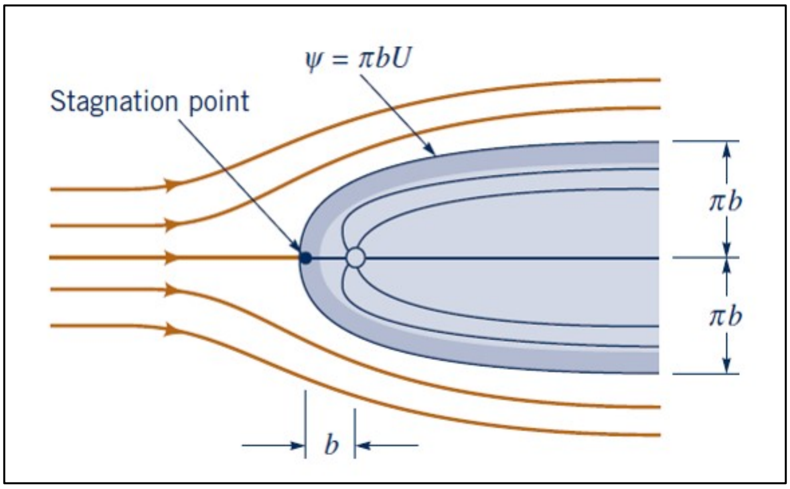



**Figure 1.21** Replacement of streamline $\psi = \pi bU$ with solid boundary to form a half-body

- **Rankine Ovals**

The half-body described in the previous section is a body that is "open" at one end. To study the flow around a closed body, a source and a sink of equal strength can be combined with a uniform flow as shown in Figure 1.22.

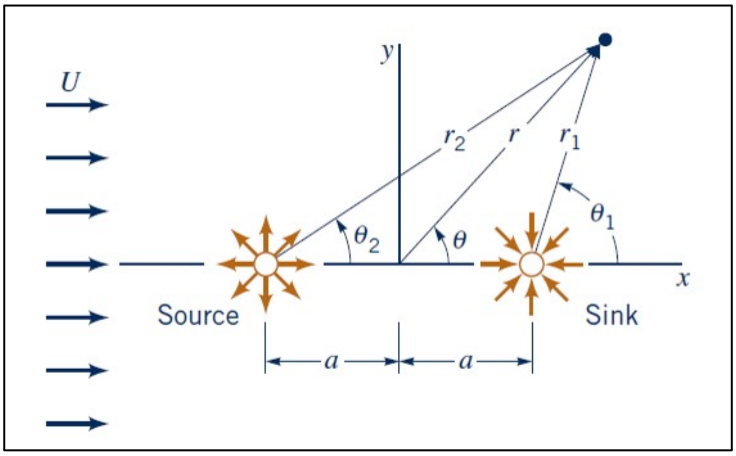



**Figure 1.22** superposition of source–sink pair and a uniform flow

The stream function and the velocity potential for this combination is:

$$\psi = Ur\,sin\,\theta + \frac{q_v}{2\pi}\left(\theta_2 - \theta_1\right)$$

$$\phi = Ur\,cos\,\theta + \frac{q_v}{2\pi}\left(ln\,r_2 - ln\,r_1\right)$$

Equivalently,

$$\psi = Ur\,sin\,\theta - \frac{q_v}{2\pi}tan^{-1}\left(\frac{2ar\,sin\,\theta}{r^2 - a^2}\right) \quad (1.36)$$

$$\phi = Ur\,cos\,\theta + \frac{1}{2}\left(\frac{q_v}{2\pi}\right)ln\left(\frac{r^2 + a^2 + 2ar\,cos\,\theta}{r^2 + a^2 - 2ar\,cos\,\theta}\right) \quad (1.37)$$

or, in Cartesian coordinates,

$$\psi = Uy - \frac{q_v}{2\pi}tan^{-1}\left(\frac{2a\,y}{x^2 + y^2 - a^2}\right) \quad (1.38)$$

$$\phi = Ux + \frac{1}{2}\left(\frac{q_v}{2\pi}\right)ln\left(\frac{(x+a)^2 + y^2}{(x-a)^2 + y^2}\right) \quad (1.39)$$

The velocity components are:

$$u = U + \frac{q_v}{2\pi}\left(\frac{x+a}{(x+a)^2 + y^2} - \frac{x-a}{(x-a)^2 + y^2}\right) \quad (1.40)$$

$$v = \frac{q_v}{2\pi}\left(\frac{y}{x^2 + (y+a)^2} - \frac{y}{x^2 + (y-a)^2}\right) \quad (1.41)$$

The stagnation points occur at:

$$x_s = \ell = \pm\left(\frac{q_v\,a}{\pi U} + a^2\right)^{1/2},\ i.e., \frac{\ell}{a} = \pm\left(\frac{q_v}{\pi U\,a} + 1\right)^{1/2}$$

$$y_s = 0$$

Where $\vec{V} = 0$ with corresponding streamline $\psi_s = 0$.

The maximum height of the Rankine oval is located at $(h, \pi/2)$ when $\psi = \psi_s = 0$, i.e.,

$$\psi = Uh - \frac{q_v}{2\pi} tan^{-1}\left(\frac{2ah}{h^2 - a^2}\right) = 0$$

or

$$\frac{h}{a} = \frac{1}{2}\left[\left(\frac{h}{a}\right)^2 - 1\right] tan\left(\frac{2\pi Ua}{q_v} \cdot \frac{h}{a}\right) \qquad (1.42)$$

which can only be solved numerically.

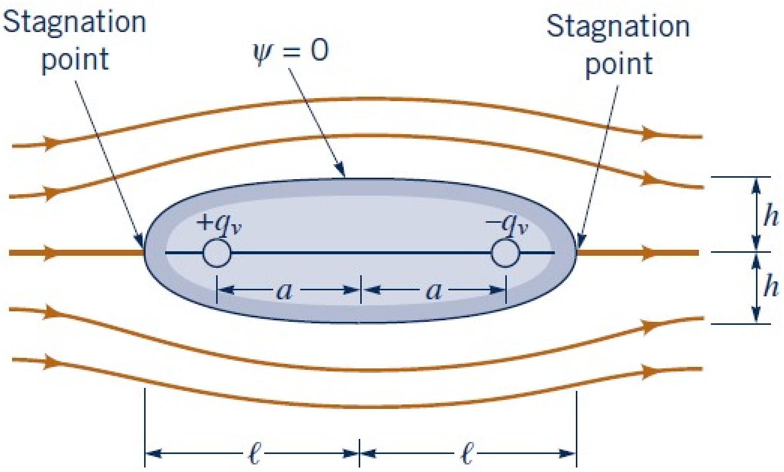



**Figure 1.23** Replacement of streamline $\psi = 0$ with solid boundary to form Rankine oval

- **Flow around a Circular Cylinder**

- **Steady cylinder**

When the distance between the source–sink pair approaches zero ($a \to 0$), the shape of the Rankine oval becomes more blunt and in fact approaches a circular shape. Since the doublet described in Section 5.2 was developed by letting a source–sink pair approach one another, it might be expected that a uniform flow in the positive $x$ −direction combined with a doublet could be used to represent flow around a circular cylinder.

At this limit with $K = \frac{q_v . a}{\pi} = const.$, the height $h$ in equation 2.42 becomes the radius of the cylinder $r_0$.

$$\frac{r_0}{a} = \frac{1}{2}\left[\left(\frac{r_0}{a}\right)^2 - 1\right] tan\left(\frac{2\pi Ua}{q_v} \cdot \frac{r_0}{a}\right) \simeq \frac{1}{2}\left[\left(\frac{r_0}{a}\right)^2 - 1\right]\left(\frac{2\pi Ua}{q_v} \cdot \frac{r_0}{a}\right)$$

$$\Rightarrow \frac{r_0}{a} = \left[\left(\frac{r_0}{a}\right)^2 - 1\right]\left(\frac{\pi U a^2}{q_v a}\frac{r_0}{a}\right)$$

$$\Rightarrow 1 = \left[\left(\frac{r_0}{a}\right)^2 - 1\right]\frac{Ua^2}{K} = \left(r_0^2 - a^2\right)\frac{U}{K} \simeq r_0^2\frac{U}{K}$$

$$r_0 = \left(\frac{K}{U}\right)^{1/2} \qquad (1.43)$$

This combination gives for the stream function and velocity potential:

$$\psi = Ur\,sin\,\theta - K\frac{sin\,\theta}{r} \qquad (1.44)$$

$$\phi = Ur\,cos\,\theta - K\frac{cos\,\theta}{r} \qquad (1.45)$$

Introducing the cylinder radius given by equation 2.43, we obtain:

$$\psi = Ur\,sin\,\theta\left(1 - \frac{K}{U}.\frac{1}{r^2}\right) = Ur\left(1 - \frac{r_0^2}{r^2}\right)sin\,\theta \quad (1.46)$$

$$\phi = Ur\,cos\,\theta\left(1 + \frac{K}{U}.\frac{1}{r^2}\right) = Ur\left(1 + \frac{r_0^2}{r^2}\right)cos\,\theta \quad (1.47)$$

The radial and circumferential velocities are:

$$v_r = U\left(1 - \frac{r_0^2}{r^2}\right)\cos\theta \quad (1.48)$$

$$v_\theta = -U\left(1 + \frac{r_0^2}{r^2}\right)\sin\theta \quad (1.49)$$

The flow net for this flow is represented in Figure 1.24.

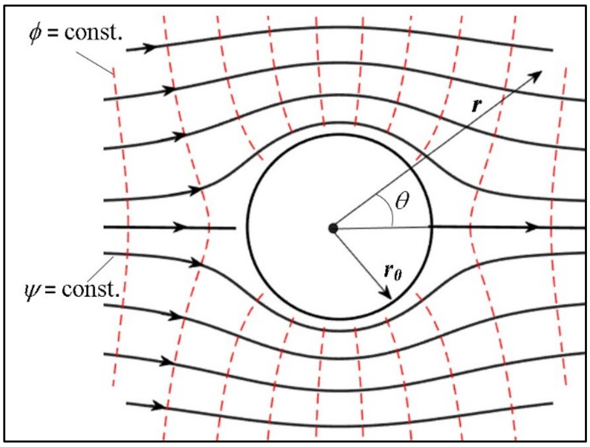



**Figure 1.24** The flow around a circular cylinder

- **Rotating cylinder**

The potential flow of a uniform parallel flow past a rotating cylinder is the superposition of a uniform parallel flow, a doublet and free vortex.

Since the vortex flow, consisting of circular streamlines, does not influence the velocity component $v_r$, the cylinder $r = r_0$ remains unchanged. Hence, the stream function and the velocity potential are given by:

$$\psi = Ur\left(1-\frac{r_0^2}{r^2}\right)\sin\theta - \frac{\Gamma}{2\pi}\ln r \qquad (1.50)$$

$$\phi = Ur\left(1+\frac{r_0^2}{r^2}\right)\cos\theta + \frac{\Gamma}{2\pi}\theta \quad (1.51)$$

The radial and circumferential velocities are given by:

$$v_r = U\left(1-\frac{r_0^2}{r^2}\right)\cos\theta \quad (1.52)$$

$$v_\theta = -U\left(1+\frac{r_0^2}{r^2}\right)\sin\theta + \frac{\Gamma}{2\pi r} \quad (1.53)$$

The tangential velocity, on the surface of the cylinder is given by:

$$v_{\theta s} = -U\left(1+\frac{r_0^2}{r^2}\right)\sin\theta + \frac{\Gamma}{2\pi r}\bigg|_{r=r_0} = -2U\sin\theta + \frac{\Gamma}{2\pi r_0}$$

We can determine the location of stagnation points on the surface of the cylinder. These points will occur at $\theta = \theta_{stag}$ where $v_q = 0$, and therefore:

$$\sin\theta_{stag} = -\frac{\Gamma}{4\pi U r_0} \qquad (1.54)$$

Several cases are possible:

If $\Gamma = 0$, then $\theta_{stag} = 0 \; or \; \pi$ —that is the stagnation points occur at the front and rear of the cylinder as are shown in Figure 2.13a.

If $-1 \leq \Gamma/4\pi U r_0 \leq 1$, the stagnation points will occur at some other location on the surface as illustrated in Figures 1.25b,c.

If the absolute value of the parameter $\Gamma/4\pi U r_0$ exceeds 1, Eq. 2.54 cannot be satisfied, and the stagnation point is located away from the cylinder as shown in Figure 1.25d.

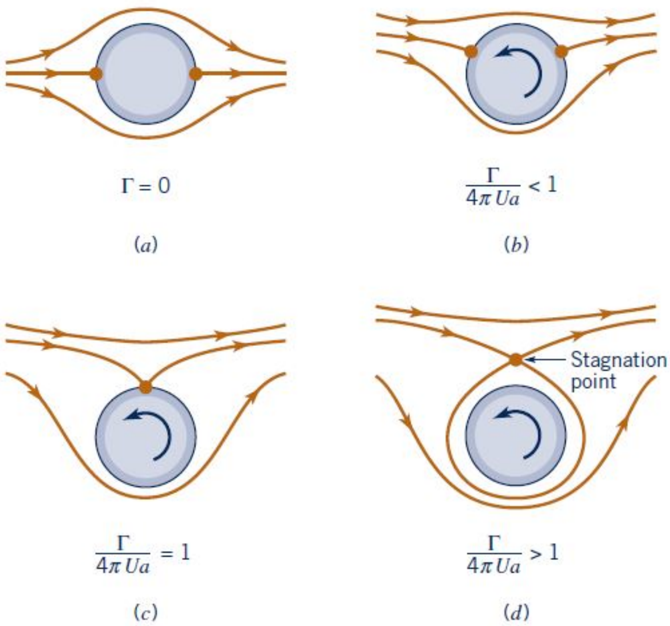



**Figure 1.25** The location of stagnation points on a circular cylinder

## 10. GRAPHICAL SUPERPOSITION METHOD

The graphical superposition method involves visually combining two potential flows by adding their stream functions and velocity potentials point by point, then plotting the resulting streamlines and equipotential lines to represent the combined flow.

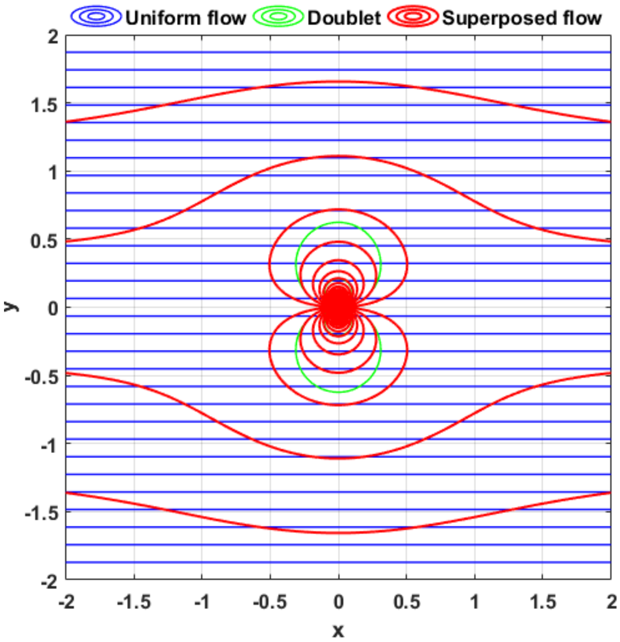
- **Steps of the graphical superposition method**

1. Plot separately the streamlines and equipotential lines of each elementary flow on the same coordinate system. (*Use graph paper or software such as MATLAB*).
2. Add the stream functions and velocity potentials of the two flows at each point in the domain.
3. Draw the level curves (contours) of the summed stream function and potential, which correspond to the streamlines and equipotentials of the superposed flow.

4. Interpret the resulting flow pattern based on the combined graphical representation.

- **MATLAB example**

**Superposition of a *uniform flow* and a *doublet* at the origin.**




```
% Parameters
U = 1;          % Uniform flow velocity
k = 5;          % Doublet strength
N = 200;        % Grid resolution
x = linspace(-2, 2, N);
y = linspace(-2, 2, N);
[X, Y] = meshgrid(x, y);

% Polar coordinates centered at origin
r = sqrt(X.^2 + Y.^2);
theta = atan2(Y, X);

% Stream function of uniform flow
psi_uniform = U * Y;

% Stream function of doublet
psi_doublet = -k ./ (2 * pi) .* sin(theta) ./ r;
psi_doublet(r < 0.01) = 0; % Avoid singularity at origin

% Superposed stream function
psi_total = psi_uniform + psi_doublet;

% Plot streamlines
figure
contour(X, Y, psi_total, 50, 'b')
hold on
contour(X, Y, psi_total, [0 0], 'r', 'LineWidth', 2) % Cylinder surface
axis equal
xlabel('x')
ylabel('y')
title('Superposition of uniform flow and doublet')
grid on
hold off
```

## 11. ELEMENTS OF COMPLEX POTENTIAL THEORY

Let $f(z)$ be a complex function defined in a domain $D \subseteq \mathbb{C}$, where $z$ is the complex number expressed as $z = x + iy$. $x$ and $y$ are real numbers and $i$ is the imaginary complex number such that $i^2 = -1$. Figure 1.26 shows the geometric representation of complex numbers.

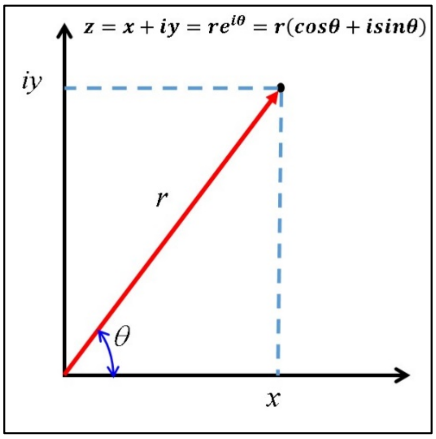



**Figure 1.26** Complex plane representation

The function $f(z) = f(x + iy) = g(x, y) + ih(x, y)$ is said to be analytic, in the domain $D$, if $f(z)$ is single valued and has a finite derivative $f'(z)$for every $z \in D$. Necessary and sufficient conditions for a function to be analytic is the following:

The four derivatives $\partial g/\partial x$, $\partial g/\partial y$, $\partial h/\partial x$, $\partial h/\partial y$ are **continuous**.

The four derivatives $\partial g/\partial x$, $\partial g/\partial y$, $\partial h/\partial x$, $\partial h/\partial y$ satisfy the **Cauchy-Riemann** equations:

$$\frac{\partial g}{\partial x} = \frac{\partial h}{\partial y} \quad \text{and} \quad \frac{\partial g}{\partial y} = -\frac{\partial h}{\partial x} \qquad (1.55)$$

Comparison of Eqs. 2.11 and 2.55 shows that the complex function $f(z) = \phi + i\psi$, with $\phi$ as the velocity potential and $\psi$ as stream function, is **analytic function** and $f$ is termed the complex potential.

From differentiation of $f(z)$, we find that:

$$\frac{df}{dz} = \frac{\partial \phi}{\partial x} + i\frac{\partial \psi}{\partial x} = \frac{\partial \phi}{i\partial y} + i\frac{\partial \psi}{i\partial y} = u - iv = (v_r - iv_\theta)e^{-i\theta} \qquad (1.56)$$

Note that to switch from cartesian coordinate velocities $(u, v)$ to polar coordinate velocities $(v_r, v_\theta)$, we apply the following transformation:

$$\begin{pmatrix} u \\ v \end{pmatrix} = \begin{bmatrix} \cos\theta & -\sin\theta \\ \sin\theta & \cos\theta \end{bmatrix} \begin{pmatrix} v_r \\ v_\theta \end{pmatrix}$$

## 12. ELEMENTARY POTENTIAL FLOWS EXPRESSED IN COMPLEX FORM

**Simple 2D flows**

Comparison of equation $f(z) = \phi + i\psi$ with the basic flows gives the following:

Uniform stream: $f(z) = U.z$

Source/Sink: $f(z) = \dfrac{q_v}{2\pi}\ln(z)$

Free vortex: $f(z) = \dfrac{-i\Gamma}{2\pi}\ln(z)$

Doublet: $f(z) = K / z$

**Superposition**

The superposition of potential flows consists simply to sum the corresponding complex potentials. The resulting complex potential represents the combined flow. For example, the flow around steady cylinder is obtained by combining uniform stream and a doublet. Hence,

$$f(z)_{\text{cylinder}} = f(z)_{\text{uniform stream}} + f(z)_{\text{doublet}}$$

$$\Rightarrow f(z) = U\,z + \frac{K}{z} = U\,z\left(1 + \frac{K/U}{z^2}\right)$$

The complex velocity is obtained by differentiating $f(z)$,

$$f'(z) = U - \frac{K}{z^2}$$

The velocity is equal to zero for all $z = (U/K)^{1/2} = r_0^2$, which represents the radius of the cylinder. Therefore,

$$f(z)=U\,z\left(1+\frac{r_0^2}{z^2}\right)$$

and, $f'(z)=U\left(1-\frac{r_0^2}{z^2}\right)$

Finally, to find the expression of $\phi, \psi, (u, v)$ $or$ $(v_r, v_\theta)$, we use the definition of the complex potential $f(z) = \phi + i\psi$ and the complex velocity $f'(z)$given by Eq. 2.56.

**Stagnation point flow**

The complex potential for this flow is given as:

$$f(z) = C.z^2$$

$$\Rightarrow f(z) = C.[x^2 - y^2 + i(2xy)]$$

$$\Rightarrow \phi = C.(x^2 - y^2) \text{ and } \psi = 2Cxy$$

Streamlines are hyperbolic lines: $y = C / x$

The complex velocity is:

$$f'(z) = 2C.z = 2C(x + iy)$$

$$\Rightarrow u = 2Cx \text{ and } v = \text{-}2Cy$$

The point (0,0) is a stagnation point, and for this reason, the flow is called flow in the vicinity of a stagnation point. The flow net for this flow is represented in Figure 1.27.

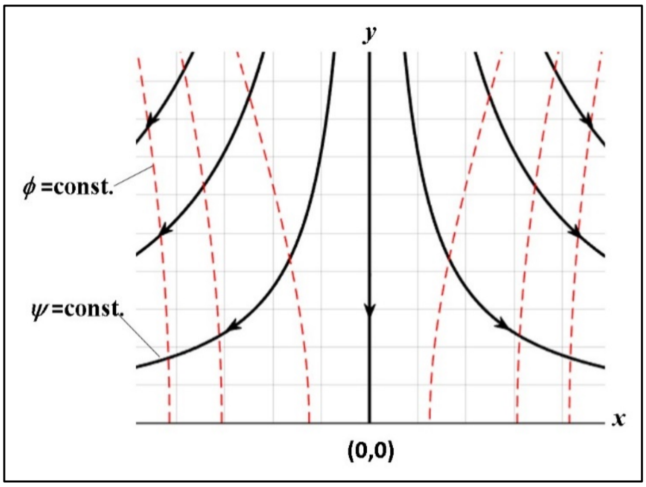



**Figure 1.27** Flow net for stagnation flow

**Flow in a sector (Flow around a corner)**

The complex potential flow in a sector or around a corner is defined by:

$$f(z) = C.z^n = Cr^n(\cos n\theta + i\sin n\theta)$$

Therefore, the velocity potential and stream function are:

$$\phi = C.r^n\cos(n\theta) \text{ and } \psi = C.r^n\sin(n\theta)$$

The complex velocity is given by:

$$f'(z) = nC.z^{n-1} = nCr^{n-1}e^{i(n-1)\theta} = (nCr^{n-1}e^{in\theta})e^{-i\theta}$$

Then, the velocity components $v_r$ and $v_\theta$ are:

$$v_r = nCr^{n-1}\cos(n\theta) \text{ and } v_\theta = -nCr^{n-1}\sin(n\theta)$$

The streamline $\psi = 0$ is obtained for $\theta = k.\pi/n$. For these values of $\theta$, we can assume the existence of a wall, $k = 0$ and $k = 1$ (i.e. $\theta = 0$ and $\theta = \pi/n$).

Several cases are possible, depending on the value of $n$:

The case n = 1 corresponds to a uniform stream parallel to a single straight boundary (Figure 1.28e).

For $n > 1$, we obtain the flow into a corner (Figures 1.28a,b,c).

For n = 2, we have flow in a region bounded by a right angle (Figure 1.28b), with streamlines in the form of rectangular hyperbole, which has already been seen in section 6.3 to be one-half of the irrotational flow near a stagnation point at a plane boundary.

Values of $n$ between $1/2$ and 1 give flow around a corner (Figures 1.28d,e,f).

The extreme case $n = 1/2$ corresponds to flow round the edge of a thin flat plate (Figure 1.28f).

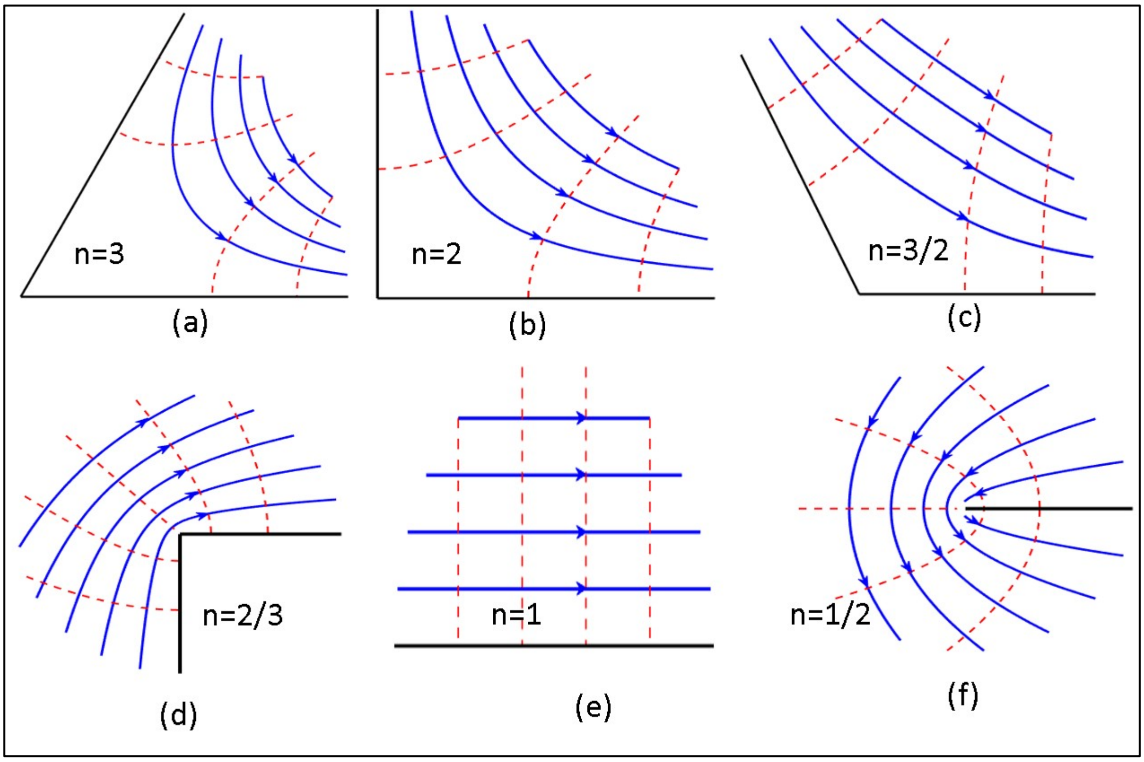



**Figure 1.28** Flow into and around a corner

We can draw the following conclusions from the formulas of $v_r$ and $v_\theta$:

For $n > 1$, the velocity magnitude $|\vec{V}| = 0$ at the vertex of the corner (i.e. this point is a stagnation point).

For $n < 1$, the velocity magnitude $|\vec{V}| \to \infty$ at the vertex of the corner.

## 13. CONFORMAL TRANSFORMATIONS METHOD

Conformal transformation or mapping is an analytic transformation between two complex planes, commonly expressed as $w = f(z)$. It has a remarkable property: it preserves the local angles between intersecting curves. This property makes conformal mapping an essential tool in fluid mechanics for converting simple flow patterns into more complex, realistic flows around various shapes.

Let lines $C_z$ and $C_z'$ in the $z$ −plane be transformations of the curves $C_w$ and $C_w'$ in the $w$ −plane, respectively (Figure 1.29). Let $\delta z$, $\delta_z^{'}$, $\delta w$, and $\delta_w^{'}$ be infinitesimal elements along the curves as shown. It can be demonstrated that, if $w = f(z)$ is an analytic function, the angles $\alpha$ and $\beta$ shown in Figure 1.29 are equal. This confirms that conformal maps keep angles intact. Consequently, the transformation of a potential flow maintains the orthogonality of the streamlines and equipotential lines, although it alters the geometry of these lines.

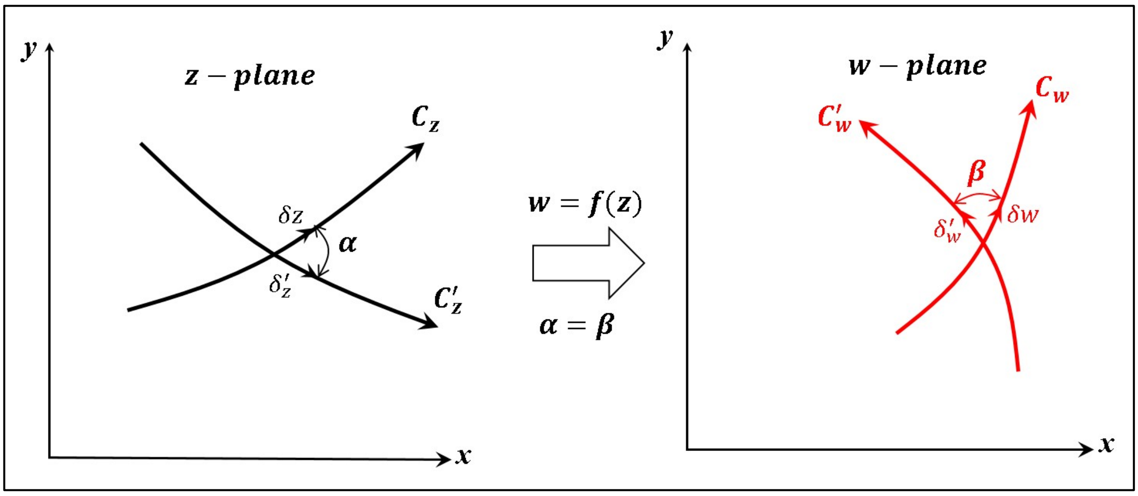



**Figure 1.29** Preservation of geometric similarity (angles) in conformal mapping

In practical fluid mechanics, conformal mapping is often applied by starting with a uniform flow $w = \phi + i\psi$ represented by a rectangular grid of equipotential and streamlines in the $w$ −plane (Figure 1.30). The conformal transformation $w = f(z)$ then converts these straight lines into curved streamlines and equipotentials in the $z$ −plane, representing the actual physical flow around objects. Figure 1.30 shows how a uniform flow with straight potential and streamlines in the $w$ −plane maps into curved lines in the $z$ −plane, which represent the real flow of interest.

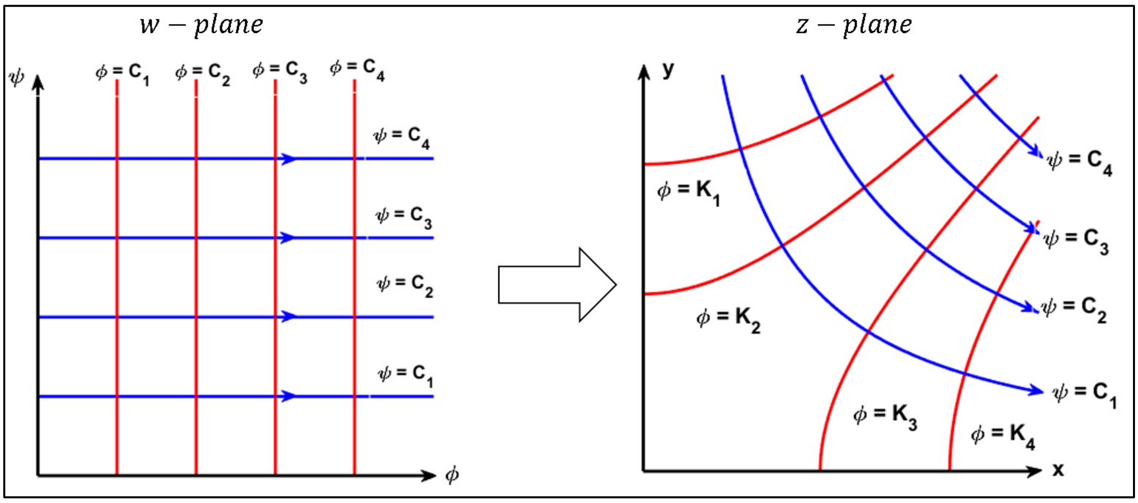



**Figure 1.30**. Flow patterns in the complex $w$ −plane and the $z$ −plane

- **Applications - Zhukhovsky transformation**

The Zhukhovsky transformation relates two complex variables $z$ and $\zeta$, and has applications in airfoil theory:

$$z = \zeta + b^2/\zeta \quad (1.57)$$

When $|\zeta\ |$ or $|z|$ is large compared to $b$, this transformation becomes an identity, so it does not change the flow condition far from the origin when moving between the $z$ and $\zeta$ planes. However, close to the origin, Eq. 2.57 transforms a circle of radius $b$ centered at the origin of the $\zeta$ −plane into a line segment on the real axis of the $z$ −plane.

To establish this, let $\zeta = b\ exp(i\theta)$ on the circle (Figure 1.31) so that Eq. 2.57 provides the corresponding point in the $z$ −plane as:

$$z = be^{i\theta} + be^{-i\theta} = 2b\cos\theta$$

As $\theta$ varies from 0 to $\pi$, $z$ goes along the $x$ −axis from $2b$ to $-2b$. As $\theta$ varies from $\pi$ to $2\pi$, $z$ goes from $-2b$ to $2b$. The circle of radius $b$ in the $\zeta$ −plane is thus transformed into a *line segment* of length $4b$ in the $z$ −plane.

The region outside the circle in the $\zeta$ −plane is mapped into the entire $z$ −plane. It can be shown that the region inside the circle is also transformed into the entire $z$ −plane. This, however, is not a concern because the interior of the circle in the $\zeta$ −plane is not important. This way, the flow around a cylinder (section 5.1) can be transformed into a flow around a thin flat plate (section 6.4f).

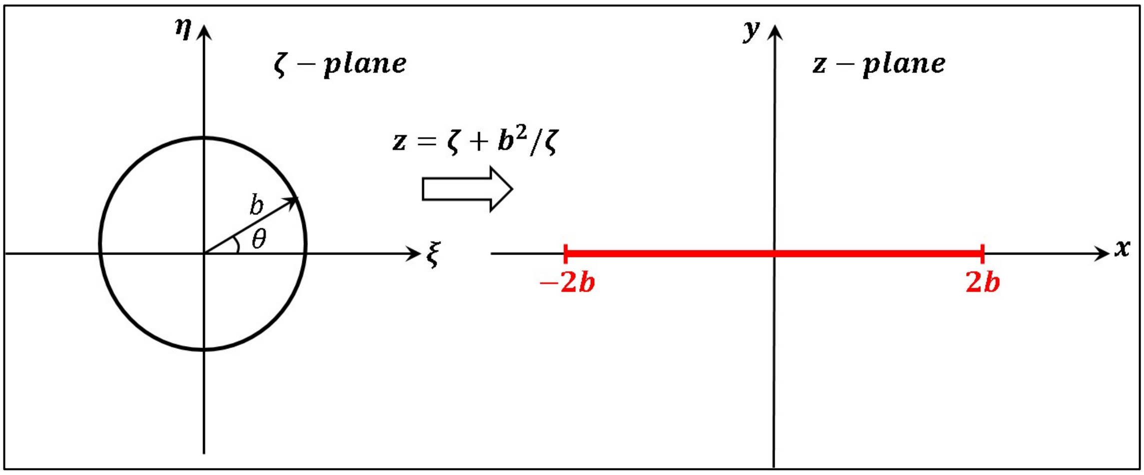



**Figure 1.31** Transformation of a circle of a radius $b$ in the $\zeta$ −plane into a line segment between $z = \pm 2b$ in the $z$ −plane

Now consider a circle of radius $a > b$ in the $\zeta$ −plane (Figure 1.32). A point $\zeta = a\ exp(i\theta)$ on this circle is transformed to:

$$z = ae^{i\theta} + \left(\frac{b^2}{a}\right)e^{-i\theta} \qquad (1.58)$$

By separating real and imaginary parts of Eq. 2.58, we obtain:

$$z = \left(a + \frac{b^2}{a}\right)cos\,\theta + i\left(a - \frac{b^2}{a}\right)sin\,\theta = x + iy$$

$$x = \left(a + \frac{b^2}{a}\right)cos\,\theta \text{ and } y = \left(a - \frac{b^2}{a}\right)sin\,\theta$$

Note that,

$$\frac{x^2}{(a + b^2/a)^2} + \frac{y^2}{(a - b^2/a)^2} = 1 \qquad (1.59)$$

For $a > b$, Eq. 2.59 represents a family of ellipses with foci at $x = \pm 2b$ (Figure 1.32) The flow around one of these ellipses (in the $z$ −plane) can be determined by first finding the flow around a circle of radius $a$ in the $\zeta$ –plane (section 5.5.2), and then using Eq. 2.57 to go to the $z$ −plane.

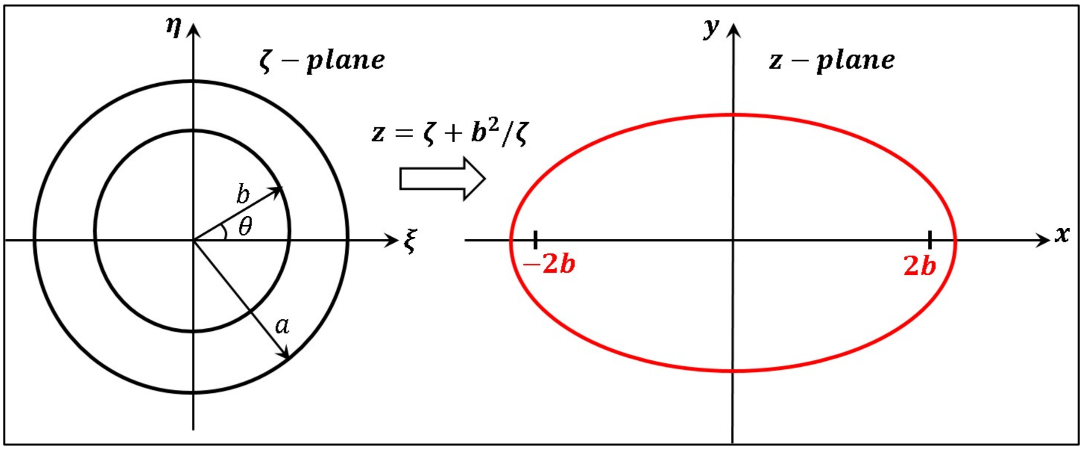



**Figure 1.32** Transformation of a circle of a radius $a$ in the $\zeta$ −plane into an ellipse in the $z$ −plane

To further illustrate this point, suppose the desired flow in the $z$ −plane is that of flow around an elliptic cylinder with *clockwise circulation* placed in a stream moving at $U$. The corresponding flow in the $\zeta$ −plane is that of flow with the same circulation around a circular cylinder of radius $a$ placed in a horizontal stream of speed $U$. The complex potential for this flow is :

$$w = U\left(\zeta + \frac{a^2}{\zeta}\right) - \frac{i\Gamma}{2\pi} ln\,\zeta \qquad (1.60)$$

The complex potential $w(z)$ in the $z$ −plane can be found by substituting the inverse of Eq. 2.57,

$$\zeta = \frac{1}{2}z + \frac{1}{2}\sqrt{z^2 - 4b^2} \ (1.61)$$

into Eq. 2.60. Here, the negative root, which falls inside the cylinder, has been excluded from Eq. 2.61.

Instead of finding the complex velocity in the $z$ −plane by directly differentiating $w(z)$, it is easier to find it using the chain rule, $u - iv = dw/dz = (dw/d\zeta\,)(d\zeta\,/dz)$. The resulting flow around an elliptic cylinder with circulation is qualitatively quite similar to that around a circular cylinder.

## 14. CHAPTER SUMMARY

The chapter on Fluid Kinematics provides a comprehensive understanding of fluid motion and introduces key concepts essential for analyzing fluid behavior. The chapter highlights the following key points:

- ***Fluid Models***: The chapter differentiates between the continuum and molecular models. The continuum model is central to this study, treating fluids as continuous media without considering molecular interactions. The behavior of fluid particles, characterized by properties like density, velocity, and pressure, is explored through this model.
- ***Eulerian vs. Lagrangian Descriptions***: Two approaches to describing fluid motion are discussed. The Eulerian description focuses on fixed locations in space to observe changes in fluid properties, while the Lagrangian description follows individual fluid particles, tracking their properties as they move. The Eulerian method is more commonly used in practical applications, although both methods are complementary.
- ***Velocity Field***: The chapter explains how fluid velocity is represented in a velocity field, with velocity components in different directions. Steady and unsteady flow are differentiated, with steady flow having a constant velocity field over time, and unsteady flow showing time-dependent changes.
- ***Streamlines, Streaklines, and Pathlines***: The concepts of streamlines, streaklines, and pathlines are introduced to describe fluid motion. Streamlines show the instantaneous direction of flow, streaklines track particles that passed through a common point, and pathlines represent the actual trajectory of a fluid particle over time. These concepts are

crucial for visualizing and analyzing the behavior of fluid flows in both steady and unsteady states.

- ***Mass Conservation & Continuity Equation***: The principle of mass conservation is discussed through the continuity equation, which expresses the conservation of mass in fluid flow. The equation ensures that mass entering and leaving a control volume remains balanced, with applications in various fluid dynamics scenarios, including steady and incompressible flows.
- ***Stream Function***: For incompressible, two-dimensional flow, the stream function is introduced as a tool to describe flow patterns. It simplifies the continuity equation and helps visualize streamlines, making it particularly useful in solving axisymmetric flows.
- ***Vorticity***: Vorticity, a key concept in fluid kinematics, quantifies the rotation of fluid particles in a flow. The chapter explains how vorticity relates to the fluid's rotational behavior, distinguishing between rotational and irrotational flow. Vorticity is computed using the velocity field and is central to understanding the intensity of rotational motion in fluids.
- ***Circulation***: Circulation, related to vorticity, measures the rotational motion in a fluid. It is defined as the line integral of the velocity along a closed curve, with nonzero circulation indicating rotational flow. The relationship between circulation and vorticity is described through Stokes' theorem
- ***Potential Flow Definition***: The chapter explains that irrotational flow, where vorticity is zero, can be described by a velocity potential function. For incompressible flow, this potential function satisfies Laplace's equation, which is central to the analysis of potential flows.
- ***Potential and Stream Functions***: The relationship between the velocity potential and stream function is described, with both satisfying Laplace's equation. The streamlines and potential lines are orthogonal, meaning they intersect at right angles. This relationship is essential for visualizing and analyzing the flow patterns.
- ***Simple 2D Potential Flows***
  - **Uniform Flow**: A flow where streamlines are parallel and the velocity is constant. It is the simplest form of flow.
  - **Source (Sink) Flow**: A radial flow emanating from (or converging toward) a point, characterized by radial streamlines and concentric equipotential lines.

- **Free Vortex**: A flow where the velocity decreases with the inverse of the radial distance from the center, forming circular streamlines.

- ***Superposition of Basic Flows***: The superposition principle is used to combine basic flow types (source, sink, vortex) to form more complex flow patterns. Examples include Rankine ovals and flows around cylinders, where the superposition of multiple potential flows can model practical situations.
  - **Flow Around a Circular Cylinder**: The flow around a cylinder is modeled by a combination of a uniform flow and a doublet (source-sink pair). This model is important in understanding aerodynamic and hydrodynamic problems.
  - **Rotating Cylinder Flow**: The potential flow of a rotating cylinder combines a uniform flow, a doublet, and a free vortex. The location of stagnation points on the cylinder surface is determined, depending on the rotation speed and flow conditions.

- ***Complex analysis of 2-D Potential Flows***: The use of complex analysis for potential flows is introduced, where complex potentials are used to describe the flow in a more analytical form. The complex potential helps in calculating velocities and visualizing the flow through streamlines and equipotential lines.

- ***Conformal Mapping***: The chapter highlights the use of conformal mapping to transform simple flow patterns into more complex shapes while preserving the essential flow characteristics. This technique is essential in fluid mechanics for modeling flow around various objects, such as airfoils, and is illustrated through the Zhukhovsky transformation.

The chapter provides a solid foundation in the theory of potential flows, emphasizing their importance in fluid dynamics. By introducing velocity potentials, stream functions, and superposition methods, it equips readers with the tools to analyze a wide range of idealized fluid flows. The use of complex potentials and conformal mapping techniques adds an analytical dimension to the study, allowing for the modeling of more complex geometries in engineering applications, such as flow around cylinders and airfoils. Through these principles, the chapter lays the groundwork for understanding and solving practical fluid flow problems in various engineering fields.

# FINITE CONTROL VOLUME ANALYSIS

## 1. INTRODUCTION

This chapter introduces the fundamentals of finite control volume analysis, a critical approach in fluid mechanics used to examine fluid behavior within a defined spatial region, whether fixed or moving. The primary objective is to apply the Reynolds Transport Theorem (RTT), mass conservation, and Newton's second law to control volumes to analyze mass flow, momentum, and angular momentum exchanges in fluid systems. Through practical examples, the chapter demonstrates how to handle steady and unsteady, compressible and incompressible flows, while addressing forces and moments exerted by fluids on interacting surfaces. This theoretical foundation is essential for understanding and designing engineering applications such as hydraulic systems, jet engines, and rotating machinery.

- **SYSTEM AND CONTROL VOLUME**

A ***system*** is a collection of matter of fixed identity (always the same atoms or fluid particles), which may move, flow, and interact with its surroundings.

A ***control volume***, on the other hand, is a volume in space (a geometric entity, independent of mass) through which fluid may flow. Unlike a system, the fluid inside a control volume can change with time.

The boundary surrounding the control volume through which the fluid passes is named the ***control surface***.

Figure 2.1 illustrates these two concepts.

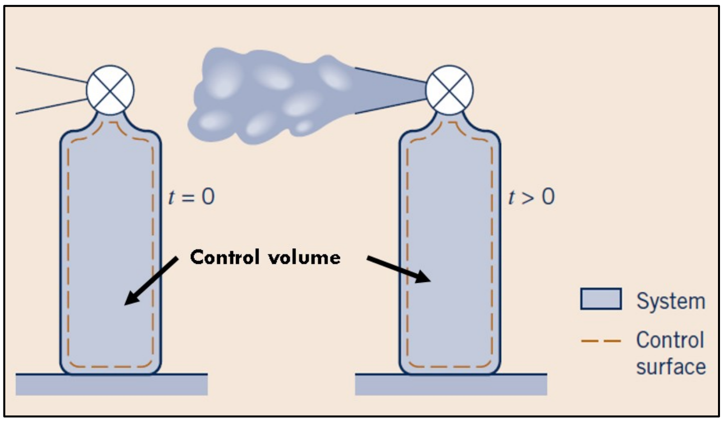



**Figure 2.1** System and control volume illustration

Different types of control volume can be defined in practice (see Figure 2.2):

Fixed control volume (stationary in space, e.g. pipe segment),

Moving control volume (e.g. jet engine moving with fluid),

Deforming control volume (changing shape/volume, e.g. expanding balloon).

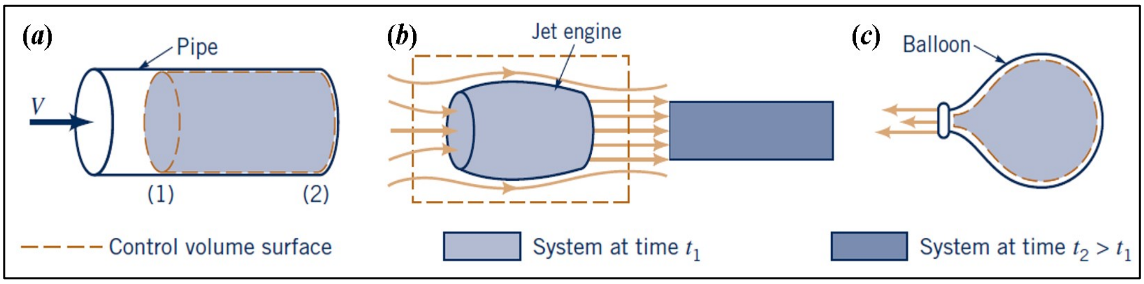



**Figure 2.2** Typical control volumes (a) fixed control volume, (b) fixed or moving control volume, (c) deforming control volume

- **THE REYNOLDS TRANSPORT THEOREM**

The purpose of the Reynolds transport theorem is to connect system analysis (fixed mass) with control volume viewpoint by relating time rates of change of fluid properties in a system to those in a control volume plus flux across boundaries.

Let $B = any\ fluid\ parameter$, such as mass, velocity, temperature, momentum, etc. Let $b = B/m$, a fluid parameter per unit mass. The mass $m$ may be that contained in a system or a control volume.

Property $B$ can be written for both the system and the control volume, as follows: System:

$$B_{sys} = \int_{sys} \rho b d\mathcal{V}$$

Control volume: $B_{cv} = \int_{cv} \rho b d\mathcal{V}$

Where $\mathcal{V}$ denotes the volume.

Remember that in general, $B_{sys} \neq B_{cv}$

**Example 3.1**

Referring again to Figure 2.1. Let $B = m$, then $b = 1$

$$\frac{dB_{sys}}{dt} = \frac{dm_{sys}}{dt} = \frac{d\left(\int_{sys} \rho d\forall\right)}{dt} = 0$$

$$\frac{dB_{cv}}{dt} = \frac{dm_{cv}}{dt} = \frac{d\left(\int_{cv} \rho d\forall\right)}{dt} < 0$$

**Derivation of the Reynolds Transport Theorem**

For simplicity's sake, let us consider the one-dimensional flow through a variable area illustrated in Figure 2.3.

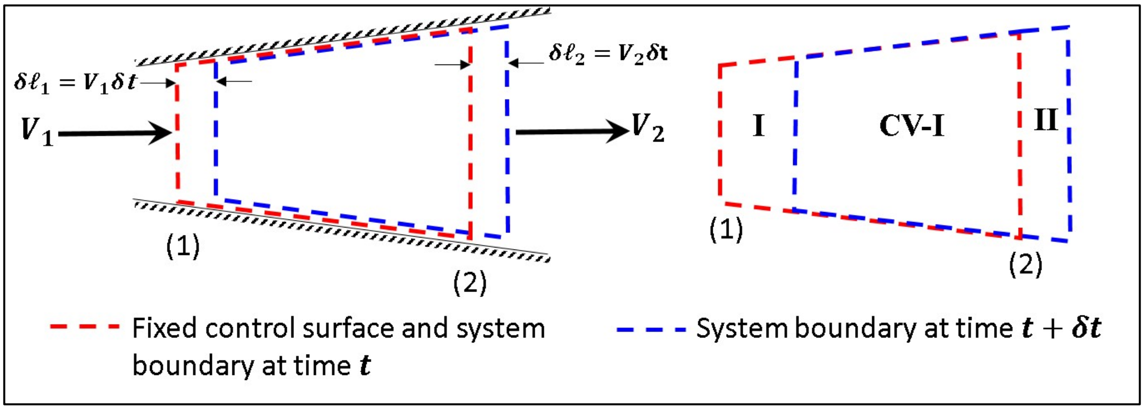



**Figure 2.3** Control volume and system for flow through a variable area pipe

At time $t$, the control volume and the system coincide. A short time later, at time $t + \delta t$, the fluid at section (1) has moved a distance $\delta\ell_1 = V_1\delta t$ to the right, where $V_1$ is the fluid velocity at section (1). Similarly, the fluid particles that coincided with section (2) of the control surface at time $t$ have moved a distance $\delta\ell_2 = V_2\delta t$ to the right, where $V_2$ is the velocity of the fluid as it passes section (2).

As is shown in Figure 3.3, the outflow from the control volume from time $t$ to $t + \delta t$ is denoted as volume **II**, the inflow as volume **I**, and the control volume itself as $CV$. Thus, the system at time $t$ consists of the fluid in section $CV$; that is "$Sys = CV$" at time $t$. At time $t + \delta t$ the system consists of the same fluid that now occupies sections $(CV - I) + II$. That is, "$Sys = CV - I + II$" at time $t + \delta t$. The control volume remains as section $CV$ for all time.

If $B$ is an extensive parameter of the system, then the value of it for the system at time $t$ is:

$$B_{sys}(t) = B_{cv}(t)$$

Its value at time $t + \delta t$ is:

$$B_{sys}(t+\delta t) = B_{cv}(t+\delta t) - B_I(t+\delta t) + B_{II}(t+\delta t)$$

Thus,

$$\frac{\delta B_{sys}}{\delta t} = \frac{B_{sys}(t+\delta t) - B_{sys}(t)}{\delta t} = \frac{B_{cv}(t+\delta t) - B_I(t+\delta t) + B_{II}(t+\delta t) - B_{sys}(t)}{\delta t}$$

By using the fact that at the initial time $t$, we have $B_{sys}(t) = B_{CV}(t)$, this expression may be rearranged as follows:

$$\frac{\delta B_{sys}}{\delta t} = \frac{B_{cv}(t+\delta t) - B_{cv}(t)}{\delta t} - \frac{B_I(t+\delta t)}{\delta t} + \frac{B_{II}(t+\delta t)}{\delta t}$$

In the limit $\rightarrow 0$ ,

$$\frac{\delta B_{sys}}{\delta t} \rightarrow \frac{DB_{sys}}{Dt}$$

$$\frac{B_{cv}(t+\delta t) - B_{cv}(t)}{\delta t} \rightarrow \frac{\partial B_{cv}}{\partial t}$$

$$B_I(t+\delta t) = (\rho_1 b_1)(\delta \cancel{V}_I) = \rho_1 b_1 A_1 V_1 \delta t$$

$$B_{II}(t+\delta t) = (\rho_2 b_2)(\delta \cancel{V}_{II}) = \rho_2 b_2 A_2 V_2 \delta t$$

$$\Rightarrow \frac{B_I(t+\delta t)}{\delta t} = \dot{B}_{in} = \rho_1 A_1 V_1 b_1$$

$$\Rightarrow \frac{B_{II}(t+\delta t)}{\delta t} = \dot{B}_{out} = \rho_2 A_2 V_2 b_2$$

Finally, we obtain:

$$\frac{DB_{sys}}{Dt} = \frac{\partial B_{cv}}{\partial t} + \dot{B}_{out} - \dot{B}_{in} \qquad (2.1)$$

or

$$\frac{DB_{sys}}{Dt} = \frac{\partial B_{cv}}{\partial t} + \rho_2 A_2 V_2 b_2 - \rho_1 A_1 V_1 b_1 \qquad (2.2)$$

This is a version of the Reynolds transport theorem valid under the following assumptions: fixed control volume with one inlet and one outlet having uniform properties (density, velocity, and the parameter $b$) across the inlet and outlet with the velocity normal to sections 1 and 2.

The simplified Reynolds transport theorem (Eq. 3.1) can be easily generalized if we give the correct interpretation to the terms $\dot{B}_{out}$ and $\dot{B}_{in}$.

In the case where the control volume comprises several inputs/outputs across the control surface (Figure 2.4), Eq.2.1 can be rewritten as follows:

$$\frac{DB_{sys}}{Dt}=\frac{\partial B_{cv}}{\partial t}+\sum\dot{B}_{out}-\sum\dot{B}_{in}=\frac{\partial B_{cv}}{\partial t}+(\dot{B}_{II\,a}+\dot{B}_{II\,b}+\dot{B}_{II\,c}+\dot{B}_{II\,d})-(\dot{B}_{I\,a}+\dot{B}_{I\,b}) \qquad (2.3)$$

Where, $\dot{B}(i)=\rho_i V_i A_i b_i$

Note that the above assumptions for equation 2.1 are also required to apply Eq.2.3.

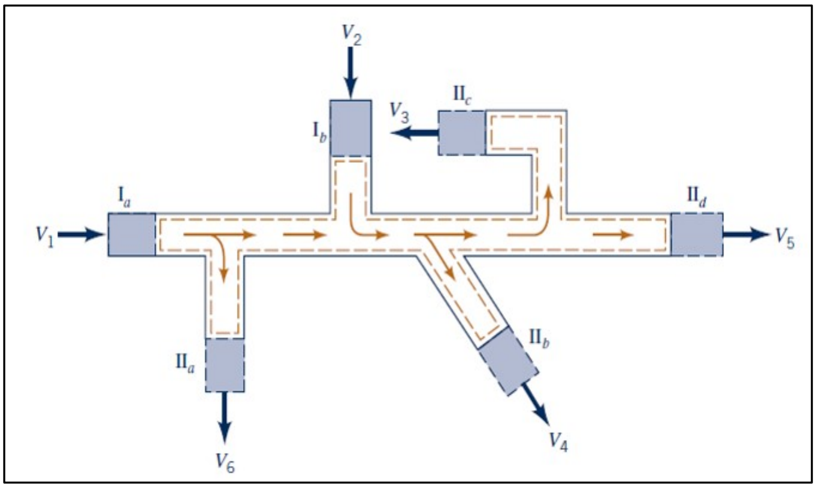



**Figure 2.4** Typical control volume with more than one inlet and outlet

Generally speaking, flows are three-dimensional, geometries are complex, and fluid properties are non-uniform. As a result, the previous assumptions are no longer valid. The flux of the fluid property $B$ through the control surface is determined in this case by integration over the surface.

For an infinitesimal area element $dA$ of the surface (see Figure 2.5), the volume of fluid crossing this element in a small time interval $\delta t$ is:

$$d\vartheta=\delta\ell\,\delta A=(\vec{V}.\vec{n}\,\delta t)\,\delta A$$

where $\vec{V}$ is the fluid velocity and $\vec{n}$ is the outward unit vector normal to the surface. The product $\vec{V}.\vec{n}$ is positive if fluid flows out and negative if it flows in.

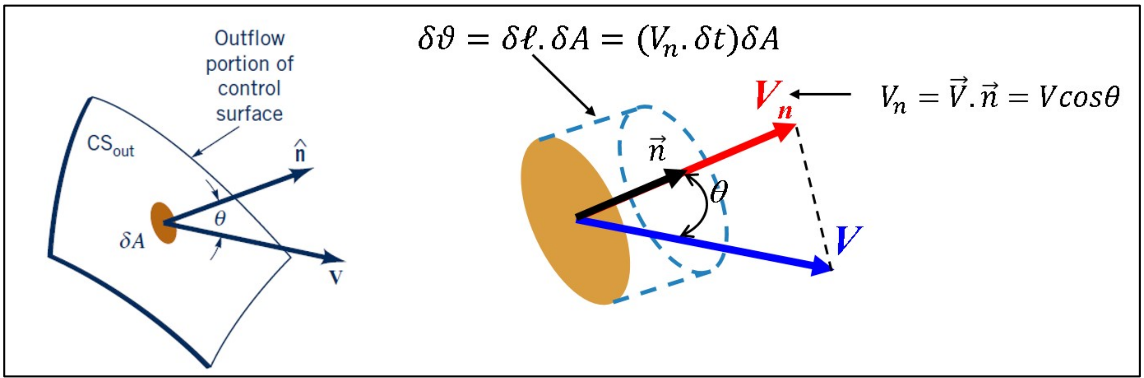



**Figure 2.5** Outflow across a portion of the control surface

The amount of the extensive property $B$ contained in this volume element is:

$$\delta B = \rho b \delta\vartheta = \rho b(\vec{V}.\vec{n}\,\delta t)\,\delta A$$

Dividing by $\delta t$ and integrating over the entire control surface gives the net rate of flow of $B$ across the surface:

$$\dot{B} = \dot{B}_{out} - \dot{B}_{in} = \int_{CS} \rho b\,\vec{V}.\vec{n}\,dA$$

Equation 3.1 is then re-expressed as:

$$\frac{DB_{sys}}{Dt} = \frac{\partial B_{cv}}{\partial t} + \int_{CS} \rho b\,\vec{V}.\vec{n}\,dA$$

This can be written in a slightly different form by using $B_{CV} = \int_{CV} \rho b\,d\vartheta$ so that:

$$\frac{DB_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{CV} \rho b\,d\vartheta + \int_{CS} \rho b\,\vec{V}.\vec{n}\,dA \quad (2.4)$$

Equation 2.4 represents the mathematical statement of the Reynolds transport theorem. It links the Lagrangian description of fluid motion, which tracks the system, to the Eulerian description based on a fixed spatial control volume. It holds for any extensive property$B$, any flow geometry, and any unsteady flow conditions.

For moving and/or deforming control volumes, Eq. 2.4 is expressed as follows:

$$\frac{DB_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{CV} \rho b \, d\vartheta + \int_{CS} \rho b \, \vec{V}_r . \vec{n} \, dA \quad (2.5)$$

Where $\vec{V_r}$ is the relative velocity $\vec{V_r} = \vec{V} - \vec{V_r}$

Note that the relative velocity is the fluid velocity relative to the moving control volume (the fluid velocity seen by an observer riding along on the control volume). The absolute velocity is the fluid velocity as seen by a stationary observer in a fixed coordinate system.

## 2. CONSERVATION OF MASS - THE CONTINUITY EQUATION

### 2.1 Derivation of the Continuity Equation

The principle of conservation of mass states that the mass of a system remains constant over time if no mass is added or removed. Mathematically, this is expressed as:

$$\frac{Dm_{sys}}{Dt} = 0$$

More generally, the mass of the system can be expressed as the integral of the density over the system volume:

$$m_{sys} = \int_{sys} \rho d\vartheta$$

Applying the Reynolds Transport Theorem with $B = m$ (mass) and $b = 1$, we obtain:

$$\frac{Dm_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{CV} \rho \, d\vartheta + \int_{CS} \rho \, \vec{V} . \vec{n} \, dA$$

From the mass conservation principle, the time rate of change of mass in the system is zero, leading to the continuity equation in integral form:

$$\frac{\partial}{\partial t}\int_{CV} \rho \, d\vartheta + \int_{CS} \rho \, \vec{V} . \vec{n} \, dA = 0 \quad (2.6)$$

This equation (*Continuity Equation*) states that the rate of increase of mass inside the control volume plus the net mass flow out of the control surface is zero, ensuring mass conservation in a fluid flow system.

### 2.2 Fixed Non-Deforming Control Volume

**Example 3.2**: **Steady, Incompressible Flow**

Seawater flows steadily through a simple conical-shaped nozzle at the end of a fire hose. If the nozzle exit velocity must be at least 20 $m/s$, determine the minimum pumping capacity required in $m^3/s$.

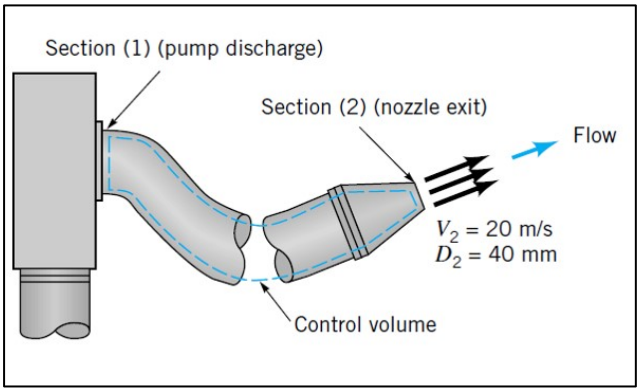



**Figure E.3.2**

**Solution**

Continuity equation: $\cancel{\frac{\partial}{\partial t}\int_{CV} \rho d\vartheta} + \int_{CS} \rho \vec{V}.\vec{n} dA = 0$ ; steady flow

$$\Rightarrow \int_{CS} \rho \vec{V}.\vec{n} dA = \dot{m}_2 - \dot{m}_1 = 0$$

$$\Rightarrow \dot{m}_2 = \dot{m}_1$$

$\Rightarrow \rho_2 Q_2 = \rho_1 Q_1$ , where $Q_1$ and $Q_2$ represent the volume flow rate at the inlet and the outlet of the nozzle respectively.

Liquid water at law speeds is considered incompressible, then $\rho_1 = \rho_2$

Consequently,

$$Q_2 = Q_1 = V_2 A_2$$

$$= V_2 . \frac{\pi D_2^2}{4} = 20 . \frac{\pi 0.04^2}{4} = 0.0251\ m^3/s$$

**Example 3.3** **Steady, Compressible Flow**

Air flows steadily between two sections in a long, straight portion of 10 cm inside diameter pipe. The uniformly distributed temperature and pressure at each section are given. The average air velocity (non-uniform velocity distribution) at section (2) is 305 m/s.

Calculate the average air velocity at section (1).

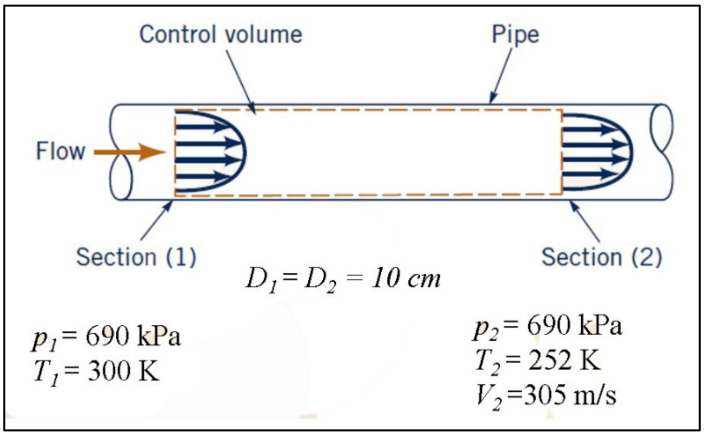



**Figure E.3.3**

**<u>Solution</u>**

Continuity equation: $\cancel{\dfrac{\partial}{\partial t}\int_{CV} \rho d\vartheta} + \int_{CS} \rho \vec{V}.\vec{n} dA = 0$ ; steady flow

$$\Rightarrow \int_{CS} \rho \vec{V}.\vec{n} dA = \dot{m}_2 - \dot{m}_1 = 0$$

$$\Rightarrow \dot{m}_2 - \dot{m}_1 = 0$$

$\Rightarrow \rho_2 \overline{V}_2 A_2 = \rho_1 \overline{V}_1 A_1$, where $\overline{V}_1$ and $\overline{V}_2$ are the average velocities at sections (1) and (2) respectively.

$$\Rightarrow \overline{V}_1 = \frac{\rho_2}{\rho_1} \overline{V}_2 \quad \text{(since } A_1 = A_2 \text{)}$$

Air at the pressures and temperatures involved in this example behaves like an ideal gas:

$$\rho = \frac{p}{RT}$$

$$\Rightarrow \overline{V}_1 = \frac{p_2 T_1 \overline{V}_2}{p_1 T_2} = \frac{124 \times 300 \times 305}{690 \times 252} = 65.3\text{m/s}$$

**<u>Example 3.4</u>** **Two Fluids**

The inner workings of a dehumidifier are shown in Figure E.3.4. Moist air (a mixture of dry air and water vapor) enters the dehumidifier at the rate of 272 $kg/hr$. Liquid water drains out of the dehumidifier at a rate of 1.36 $kg/hr$. Determine the mass flowrate of the dry air and the water vapor leaving the dehumidifier.

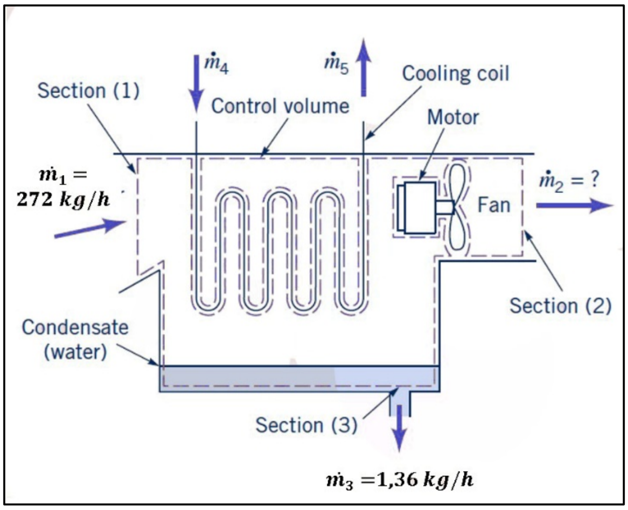



**Figure E3.4**

**<u>Solution</u>**

The contents of the control volume are the air and water vapor mixture and the condensate (liquid water) in the dehumidifier at any instant.

Not included in the control volume are the fan and its motor, and the condenser coils and refrigerant.

Continuity equation: $\cancel{\dfrac{\partial}{\partial t}\int_{CV} \rho d\vartheta} + \int_{CS} \rho \vec{V}.\vec{n}dA = 0$ ; steady flow

$$\Rightarrow \int_{CS} \rho \vec{V}.\vec{n}dA = \dot{m}_2 + \dot{m}_3 - \dot{m}_1 = 0$$

$$\Rightarrow \dot{m}_2 = \dot{m}_1 - \dot{m}_3$$

$$\Rightarrow \dot{m}_2 = 272 - 1.36 = 270.64\ kg/h$$

**<u>Example 3.5</u>** **Non-uniform Velocity Profile**

Incompressible, laminar water flow develops in a straight pipe having radius $R$ as indicated in Figure E3.5. At section (1), the velocity profile is uniform; the velocity is equal to a constant value $U$ and is parallel to the pipe axis everywhere. At section (2), the velocity profile is axisymmetric and parabolic, with zero velocity at the pipe wall and a maximum value of $u_{max}$ at the centerline.

Figure E.2.5

(a) How are $U$ and $u_{max}$ related ?

(b) How are the average velocity $V_2$ at section (2), and $u_{max}$ related ?

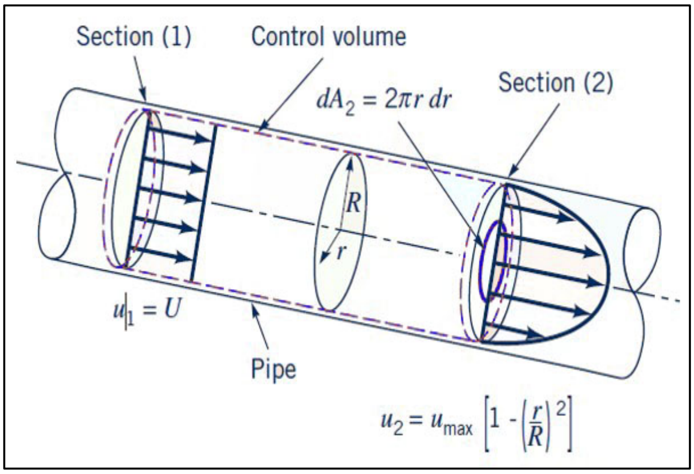



**Figure E3.5**

<u>**Solution**</u>

Continuity equation: $\cancel{\dfrac{\partial}{\partial t}\int_{CV} \rho d\vartheta} + \int_{CS} \rho \vec{V}.\vec{n} dA = 0$ ; steady flow

$$\int_{(1)} \rho \vec{V}.\vec{n} dA = -\rho_1 A_1 U$$

$$\int_{(2)} \rho \vec{V}.\vec{n} dA = \int_0^R \rho_2 u_{\max} \left[1 - \left(\frac{r}{R}\right)^2\right] .2\pi r dr$$

$$= 2\pi \rho_2 u_{\max} \int_0^R \left[r - \frac{r^3}{R^2}\right] .dr$$

$$= \frac{\pi \rho_2 u_{\max} R^2}{2}$$

$$\Rightarrow -\rho_1 A_1 U + \frac{\pi \rho_2 u_{max} R^2}{2} = 0$$

$A_1 = \pi R^2$ and $\rho_1 = \rho_2$

$$\boxed{\Rightarrow u_{max} = U}$$

The average velocity can be determined as follows:

$$\rho_2 A_2 V = \int_{(2)} \rho \vec{V}.\vec{n} dA$$

$$\Rightarrow \rho_2 V_2 \pi R^2 = \frac{\pi \rho_2 u_{max} R^2}{2}$$

$$\Rightarrow u_{max} = \frac{U}{2}$$

**Example 3.6** **Unsteady Flow**

A bathtub is being filled with water from a faucet. The rate of flow from the faucet is steady at $20\ l/min$. The tub volume is approximated by a rectangular space as indicated in Figure E.3.6.

Estimate the time rate of change of the depth of water in the tub, in in/min at any instant.

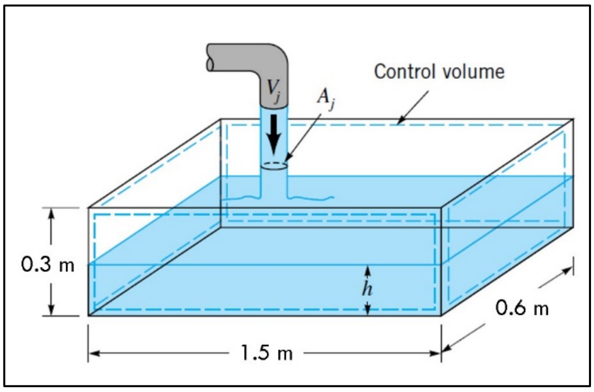



**Figure E3.6**

**Solution**

Continuity equation for all the control volume:

$$\frac{\partial}{\partial t} \int_{\substack{air \\ volume}} \rho_{air} d\vartheta_{air} + \frac{\partial}{\partial t} \int_{\substack{water \\ volume}} \rho_{water} d\vartheta_{water} - \dot{m}_{water} + \dot{m}_{air} = 0$$

Mass conservation for air:

$$\frac{\partial}{\partial t} \int_{\substack{air \\ volume}} \rho_{air} d\vartheta_{air} + \dot{m}_{air} = 0$$

Mass conservation for water:

$$\frac{\partial}{\partial t}\int_{\substack{water\\volume}} \rho_{water}\, d\vartheta_{water} - \dot{m}_{water} = 0$$

$$\Rightarrow \frac{\partial}{\partial t}\int_{\substack{water\\volume}} \rho_{water}\, d\vartheta_{water} = \dot{m}_{water}$$

The first term of the previous equation can be expressed as (see figure E3.6):

$$\frac{\partial}{\partial t}\int_{\substack{water\\volume}} \rho_{water}\, d\vartheta_{water} = \frac{\partial}{\partial t}\left\{\rho_{water}\left[h(1.5)(0.6)+(0.3-h)A_j\right]\right\}$$

Thus,

$$\rho_{water}\left(0.9-A_j\right)\frac{\partial h}{\partial t} = \dot{m}_{water} = \rho_{water} Q_{water}$$

$$\Rightarrow \frac{\partial h}{\partial t} = \frac{Q_{water}}{\left(0.9-A_j\right)}$$

$A_j \ll 0.99\ m^2$, hence

$$\frac{\partial h}{\partial t} = \frac{Q_{water}}{0.9} = \frac{20\times10^{-3}}{0.9} = 0.0222\ \text{m/min} = 2.22\ \text{cm/min}$$

### 2.3 Moving Non-Deforming Control Volume

**<u>Example 3.7</u>** **Compressible Flow with a Moving Control Volume**

An airplane moves forward at a speed of $971\ km/hr$ as shown in Figure E3.7. The frontal intake area of the jet engine is $0.80\ m^2$ and the entering air density is $0.736\ kg/m^3$. A stationary observer determines that relative to the Earth, the jet engine exhaust gases move away from the engine with a speed of $1050\ km/hr$. The engine exhaust area is $0.588\ m^2$, and the exhaust gas density is $0.515\ kg/m^3$.

Estimate the mass flowrate of fuel into the engine in $Kg/hr$.

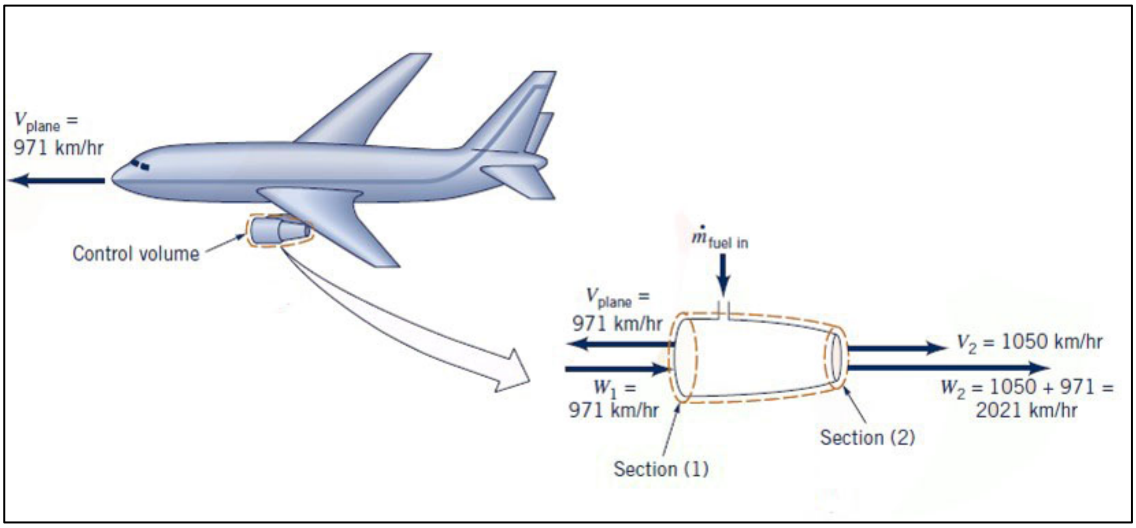



**Figure E3.7**

**Solution**

Continuity equation: $\frac{\partial}{\partial t}\cancel{\int_{CV} \rho d\vartheta} + \int_{CS} \rho \vec{W}.\vec{n}dA = 0$; steady flow

$$\Rightarrow \int_{CS} \rho \vec{W}.\vec{n}dA = -\dot{m}_{fuel\_in} - \rho_1 A_1 W_1 + \rho_2 A_2 W_2 = 0$$

or, $\dot{m}_{fuel\_in} = \rho_2 A_2 W_2 - \rho_1 A_1 W_1$

$$\vec{W}_1 = \vec{V}_{plane} \quad \text{and} \quad \vec{V}_2 = \vec{W}_2 + \vec{V}_{plane}$$

or, $W_1 = V_{plane}$ and $W_2 = V_2 + V_{plane}$

$W_1 = 971$ km/h and $W_2 = 1050 + 971 = 2021$ km/h

Hence,

$$\dot{m}_{fuel\_in} = (0.515)(0.558)(2021\times10^3) - (0.736)(0.80)(971\times10^3) = 9100 \text{ kg/h}$$

**Example 3.8** **Relative velocity**

Water enters a rotating lawn sprinkler through its base at the steady rate of 1000 $ml/s$ as sketched in Figure E3.8. The exit area of each of the two nozzles is 30 $mm^2$.

Determine the average speed of the water leaving the nozzle, relative to the nozzle, if the rotary sprinkler head is stationary, the sprinkler head rotates at 600 rpm, and

the sprinkler head accelerates from 0 to 600 rpm.

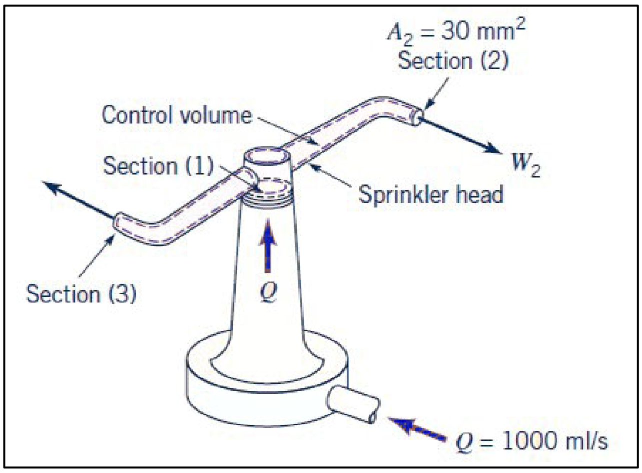



**Figure E3.8**

**Solution**

Continuity equation: $\cancel{\frac{\partial}{\partial t}\int_{CV} \rho d\vartheta} + \int_{CS} \rho \vec{W}.\vec{n} dA = 0$; steady flow

$$\Rightarrow \quad \rho_2 A_2 W_2 + \rho_3 A_3 W_3 - \rho_1 A_1 W_1 = 0$$

$$\rho_1 = \rho_2 = \rho_3 \;\; ; \;\; A_2 = A_3 \;\; ; \;\; W_2 = W_3 \;\; ; \;\; \text{and} \quad Q = A_1 W_1$$

$$\Rightarrow W_2 = \frac{Q}{2A_2}$$

$$W_2 = \frac{1000 \times 10^{-3} \times 10^{-3}}{2 \times 30 \times 10^{-6}} = 16.7 \text{ m/s}$$

and **(c)**, the value of $W_2$ is independent of the speed of rotation of the sprinkler head and represents the average velocity of the water exiting from each nozzle with respect to the nozzle for cases (a), (b), and (c).

### 2.4 Deforming Control Volume

**Example 3.9** **Deforming Control Volume**

A syringe (Figure E3.9) is used to inoculate a cow. The plunger has a face area of 500 $mm^2$. The liquid in the syringe is to be injected steadily at a rate of 300 $cm^3/min$.

The leakage rate past the plunger is 0.10 times the volume flow rate out of the needle.

With what speed should the plunger be advanced?

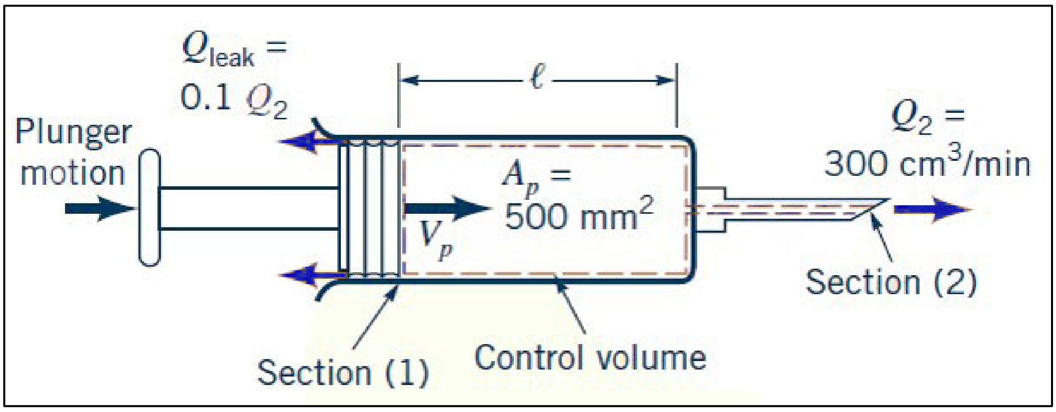



**Figure E3.9**

__Solution__

Continuity equation: $\frac{\partial}{\partial t}\int_{CV}\rho d\vartheta + \int_{CS}\rho\vec{W}.\vec{n}dA = 0$

$$\Rightarrow \frac{\partial}{\partial t}\int_{CV}\rho d\vartheta + \dot{m}_2 + \rho Q_{leak} = 0$$

$$\int_{CV}\rho d\vartheta = \rho(\ell A_p + \vartheta needle)$$

$$\Rightarrow \frac{\partial}{\partial t}\int_{CV}\rho d\vartheta = \rho A_p\frac{\partial\ell}{\partial t}$$

Note that, $V_p = -\frac{\partial\ell}{\partial t}$ (speed of the piston)

$$\Rightarrow -\rho A_p V_p + \dot{m}_2 + \rho Q_{leak} = 0$$

$$\Rightarrow -\rho A_p V_p + \rho Q_2 + \rho Q_{leak} = 0$$

$$\Rightarrow V_p = \frac{Q_2 + Q_{leak}}{A_p} = \frac{1.1 Q_2}{A_p}$$

$$V_p = \frac{1.1(300\times10^{-6}/60)}{500\times10^{-6}} = 0.011 \text{ m/s} = 660 \text{ mm/min}$$

# 3. NEWTON'S SECOND LAW - THE LINEAR MOMENTUM AND MOMENT-OF-MOMENTUM EQUATIONS

## 3.1 Derivation of the Linear Momentum Equation

Newton's Second Law, for a system, states that the time rate of change of the linear momentum of a system equals the sum of the external forces acting on it.

Since momentum is mass times velocity, the momentum of a small particle of mass $dm = \rho d\vartheta$ is $\vec{V}.\rho d\vartheta$. Thus, the momentum of the entire system is $\int_{sys} \vec{V}.\rho d\vartheta$ and Newton's law becomes:

$$\frac{D}{Dt}\int_{CV} \vec{V}\rho d\vartheta = \sum \vec{F}_{sys} \quad (2.7)$$

When a control volume is coincident with a system at an instant of time, the forces acting on the system and the forces acting on the contents of the coincident control volume(see Figure 2.6) are instantaneously identical, that is,

$$\sum \vec{F}_{sys} = \sum \vec{F}_{\substack{\text{contents of the}\\ \text{coincident control volume}}} \quad (2.8)$$

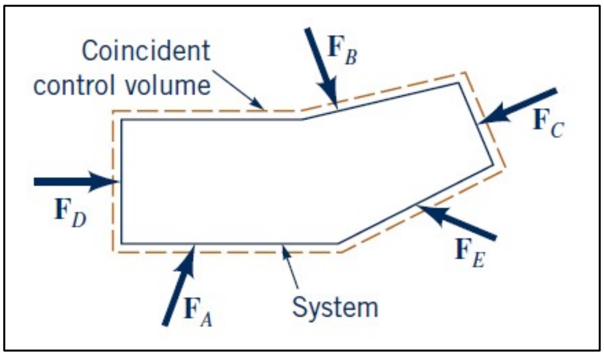



**Figure 2.6** External forces acting on system and coincident control volume

For a system and the contents of a coincident control volume that is fixed and non-deforming, the Reynolds transport theorem [with $b$ set equal to the velocity (i.e., momentum per unit mass), and $B_{sys}$ being the system momentum] allows us to conclude that:

$$\frac{D}{Dt}\int_{sys} \vec{V}\rho d\vartheta = \frac{\partial}{\partial t}\int_{CV} \vec{V}\rho d\vartheta + \int_{CS} \vec{V}\rho\vec{V}.\vec{n}\, dA \quad (2.9)$$

From Eqs. 3.7, 3.8 and 3.9 we conclude that:

$$\boxed{\frac{\partial}{\partial t}\int_{CV} \vec{V}\rho\, d\vartheta + \int_{CS} \vec{V}\rho\vec{V}.\vec{n}\, dA = \sum \vec{F}_{\substack{\text{contents of the}\\ \text{coincident control volume}}}} \qquad (2.10)$$

Equation 3.10 represents the ***linear momentum equation***.

*Note that Eq. 3.10 is a vector equation, so it can be decomposed in the three directions $x, y$ and $z$.*

### 3.2 Application of the Linear Momentum Equation

**Example 3.10** **Change in Flow Direction**

As shown in Figure E3.10a, a horizontal jet of water exits a nozzle with a uniform speed of $V_1 = 3.05\ m/s$, strikes a vane, and is turned through an angle $\theta$.

Determine the anchoring force needed to hold the vane stationary if gravity and viscous effects are negligible.

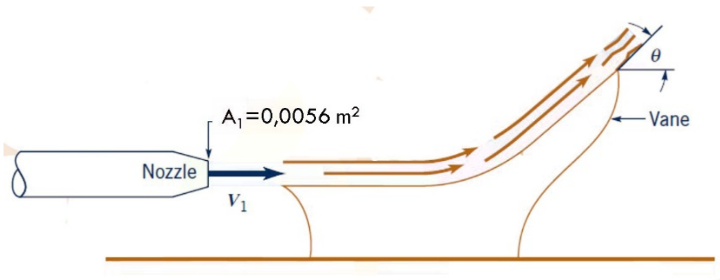



**Figure E3.10a**

**Solution**

We select a control volume that includes the vane and a portion of the water (see Figure. E3.10b) and apply the linear momentum equation to this fixed control volume:

$$\cancel{\frac{\partial}{\partial t}\int_{CV} \vec{V}\rho\, d\vartheta} + \int_{CS} \vec{V}\rho\vec{V}.\vec{n}\, dA = \sum \vec{F} \text{ ; Steady flow}$$

Hence, the $x$ and $z$ components of Eq. 3.10 become:

$$\int_{CS} u\rho\vec{V}.\vec{n}\, dA = \sum F_x$$

$$\int_{CS} w\rho\vec{V}.\vec{n}\,dA = \sum F_z$$

or,

$$u_2\rho A_2V_2 - u_1\rho A_1V_1 = \sum F_x$$

$$w_2\rho A_2V_2 - w_1\rho A_1V_1 = \sum F_z$$

With negligible gravity and viscous effects, and since $p_1 = p_2$, the speed of the fluid remains constant so that $V_1 = V_2 = 3.05\ m/s$ (Bernoulli equation). Hence, at section (1), $u_1 = V_1$, $w_1 = 0$, and at section (2), $u_2 = V_1 cos\theta$ , $w_2 = V_1\ sin\theta$. Thus,

$$V_1\cos\theta\,\rho A_2V_1 - V_1\rho A_1V_1 = F_{Ax}$$

$$V_1\sin\theta\,\rho A_2V_1 - (0)\rho A_1V_1 = F_{Az}$$

Conservation of mass states that for this incompressible flow $A_1V_1 = A_2V_2$ or $A_1 = A_2$ since $V_1 = V_2$. Consequently,

$$\mathrm{F}_{Ax} = -\rho A_1V_1^2 + \rho A_1V_1^2\cos\theta = -\rho A_1V_1^2(1-\cos\theta)$$
$$\mathrm{F}_{Az} = \rho A_1V_1^2\sin\theta$$

$$\Rightarrow\quad F_{Ax} = -(998)(0.0056)(3.05)^2(1-cos\,\theta) = -52(1-cos\,\theta)$$
$$F_{Az} = (998)(0.0056)(3.05)^2\,sin\,\theta = 52\,sin\,\theta$$

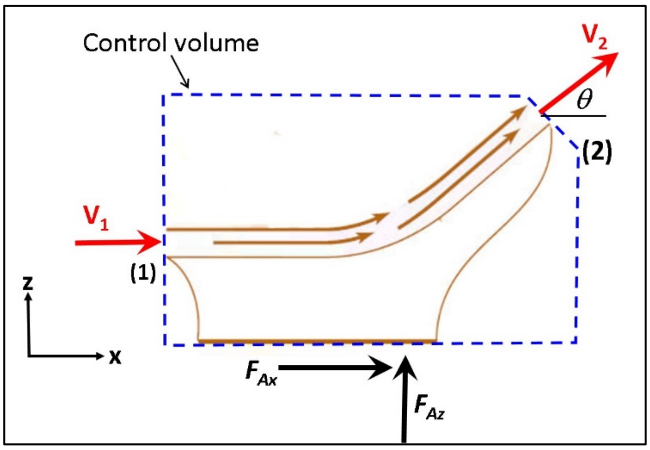



**Figure E3.10b**

**<u>Example 3.11</u>** **Pressure and Change in Flow Direction**

Water flows through a horizontal pipe bend as shown in Figure E3.11. The flow cross-sectional area is constant at a value of 0.0093 $m^2$ through the bend. The magnitude of the flow velocity everywhere in the bend is axial and 15.24 $m/s$. The absolute pressures at the entrance and exit of the bend are 207 $kPa$ and 165 $kPa$, respectively.

Calculate the horizontal (x and y) components of the anchoring force required to hold the bend in place.

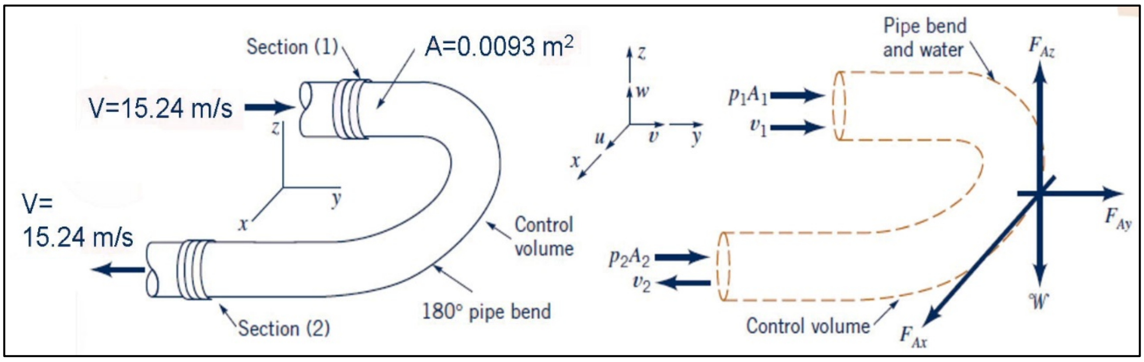



**Figure E3.11**

**<u>Solution</u>**

Linear momentum equation: $\cancel{\frac{\partial}{\partial t}\int_{CV}\vec{V}\rho d\vartheta} + \int_{CS}\vec{V}\rho\vec{V}.\vec{n}\,dA = \sum\vec{F}$ ; Steady flow

$$\Rightarrow \begin{cases} \int_{CS} u\rho\vec{V}.\vec{n}\,dA = F_{Ax} \\ \int_{CS} v\rho\vec{V}.\vec{n}\,dA = F_{Ay} \end{cases}$$

$F_{Ax} = 0$, since $u = 0$ at both section (1) and (2). Hence,

$$\int_{CS} v\rho\vec{V}.\vec{n}\,dA = F_{Ay} + p_1A_1 + p_2A_2$$

$$\Rightarrow (+v_1)(-\dot{m}_1) + (-v_2)(+\dot{m}_2) = F_{Ay} + p_1A_1 + p_2A_2$$

$$\dot{m}_1 = \dot{m}_2 = \dot{m}$$

$$\Rightarrow\ -\dot{m}(v_1 + v_2) = F_{Ay} + p_1A_1 + p_2A_2$$

$$\Rightarrow\ \ F_{Ay} = -\dot{m}(v_1 + v_2) - p_1A_1 - p_2A_2$$

$$\dot{m} = \rho_1 v_1 A_1 = (998)(15.24)(0.0093) = 141.4 \text{ kg/s}$$

$$F_{Ay} = -(141.4)(15.24+15.24) - (107)(0.0093) - (65)(0.0093) = \ -5909 \text{ N}$$

**Example 3.12** **Thrust**

A static thrust stand as sketched in Figure E3.12 is to be designed for testing a jet engine. The following conditions are known for a typical test: Intake air velocity = $200\ m/s$; exhaust gas velocity = $500\ m/s$; intake cross-sectional area = $1\ m^2$; intake static pressure = $-22{,}5\ kPa\ =\ 78{,}5\ kPa\ (abs)$; intake static temperature = $268\ K$; exhaust static pressure = $0\ kPa\ =\ 101\ kPa\ (abs)$.

Estimate the nominal anchoring force for which to design.

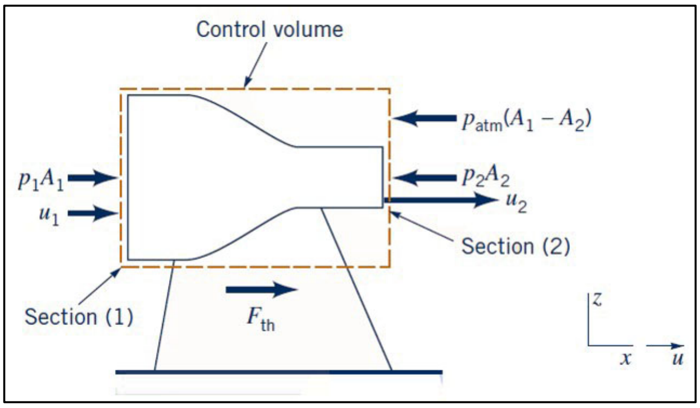



**Figure E3.13**

**Solution**

Linear momentum equation: $\cancel{\dfrac{\partial}{\partial t}\int_{CV} \vec{V}\rho d\vartheta} + \int_{CS} \vec{V}\rho\vec{V}.\vec{n}\, dA = \sum \vec{F}$ ; Steady flow

$$\sum \vec{F} = p_1 A_1 + F_{th} - p_2 A_2 - p_{atm}(A_1 - A_2)$$

$$\int_{CS} \vec{V}\rho\vec{V}.\vec{n}\, dA = (+u_1)(-\dot{m}_1) + (+u_2)(+\dot{m}_2)$$

$$\Rightarrow \quad (+u_1)(-\dot{m}_1) + (+u_2)(+\dot{m}_2) = (p_1 - p_{atm})A_1 - (p_2 - p_{atm})A_2 + F_{th}$$

Conservation of mass: $\dot{m} = \dot{m}_1 = \rho_1 A_1 u_1 = \dot{m}_2 = \rho_2 A_2 u_2$

$$\Rightarrow \quad \dot{m}(u_2 - u_1) = p_1 A_1 - p_2 A_2 + F_{th} \ \ (p_1, p_2 \text{ are gage pressures})$$

$$\Rightarrow \quad F_{th} = -p_1 A_1 + p_2 A_2 + \dot{m}(u_2 - u_1)$$

To calculate $F_{th}$ we need to determine the mass flowrate $\dot{m} = \rho_1 \, A_1 \, u_1$.

From the ideal gas equation of state:

$$\rho_1 = \frac{p_1}{RT_1} = \frac{78.5\times10^3}{286.9\times268} = 1.02 \text{ kg/m}^3 \Rightarrow \quad \dot{m} = \rho_1 A_1 u_1 = 1.02\times1\times200 = 204 \text{ kg/s}$$

$$\Rightarrow \quad F_{th} = -(-22.5\times10^3)(1) + 204(500-200) = 22500 + 61200 = 83700 \text{ N}$$

**<u>Example 3.13</u>** **Moving Control Volume**

A vane on wheels moves with constant velocity $V_0$ when a stream of water, having a nozzle exit velocity of $V_1$, is turned 45° by the vane as indicated in Figure E3.13. The speed of the water jet leaving the nozzle is 30 $m/s$, and the vane is moving to the right with a constant speed of 6 $m/s$.

Determine the magnitude and direction of the force, $R$, exerted by the stream of water on the vane surface.

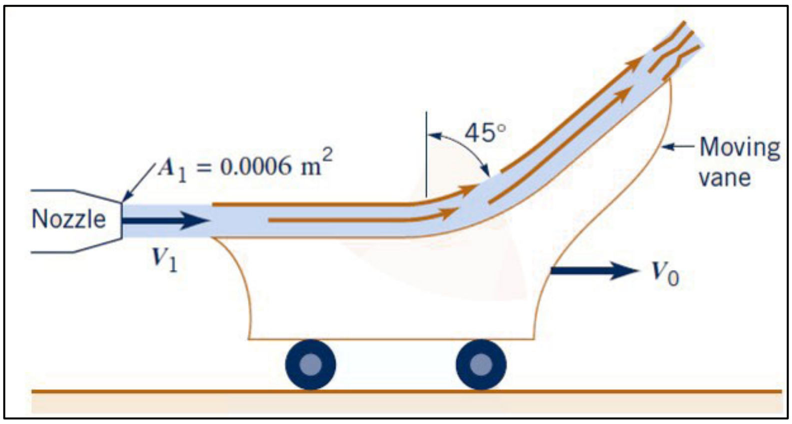
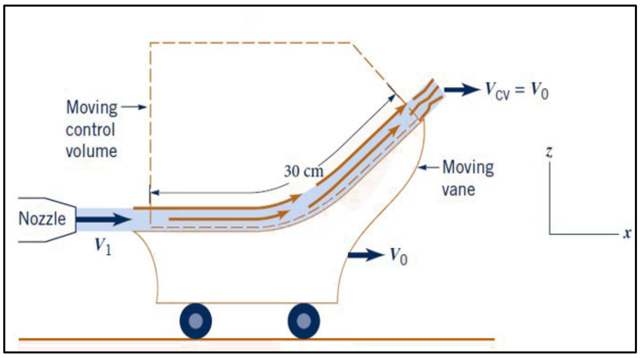



**Figure E3.13a**

**<u>Solution</u>**

The control volume is chosen as shown in figure E3.13b. The components Rx and RZ of the force R exerted by the water on the blade are shown in figure E13.c.



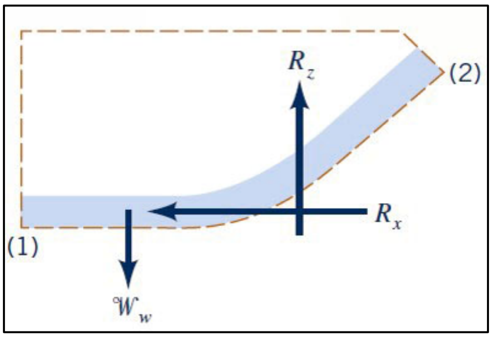




**Figure E3.13b** **Figure E3.13c**

Linear momentum equation: $\frac{\partial}{\partial t}\cancel{\int_{CV} \vec{V}\rho d\vartheta} + \int_{CS} \vec{V}\rho\vec{V}.\vec{n}\, dA = \sum \vec{F}$ ; Steady flow

$$\int_{CS} W_x \rho \vec{W}.\vec{n}\, dA = -R_x$$

$$\int_{CS} W_z \rho \vec{W}.\vec{n}\, dA = R_z - \mathcal{W}_w$$

Note that the velocity of the fluid relative to the control volume is used here.

$$\Rightarrow \begin{cases} (+W_1)(-\dot{m}_1) + (+W_2 \cos 45°)(+\dot{m}_2) = -R_x \\ (+W_2 \sin 45°)(+\dot{m}_2) = R_z - \mathcal{W}_w \end{cases}$$

$\dot{m}_1 = \rho_1 W_1 A_1$ and $\dot{m}_2 = \rho_2 W_2 A_2$

We assume that the water flow is frictionless and that the change in water elevation across the vane is negligible. Hence,

$$W_1 = W_2 = V_1 - V_0 = 30 - 6 = 24 \text{ m/s}$$

The conservation of mass is written as:

$\dot{m}_1 = \rho_1 W_1 A_1 = \dot{m}_2 = \rho_2 W_2 A_2$, with $\rho_1 = \rho_2 = 1000 \text{ kg/m}^3$

$$\Rightarrow \begin{cases} R_x = \rho W_1^2 A_1 (1 - \cos 45°) \\ R_z = \rho W_1^2 (\sin 45°) A_1 + \mathcal{W}_w \end{cases}$$

$$\mathcal{W}_w = \rho g A_1 \ell = 1000 \times 9.81 \times 0.0006 \times 0.3 = 1.77 \text{ N}$$

$$R_x = 1000 \times 24^2 \times 0.0006 (1 - \cos 45°) = 101.2 \text{ N}$$

$R_z = 1000 \times 24^2 (\sin 45°) 0.0006 + 1.77 = 246.1 \text{ N}$

The $\vec{R}$ modulus is: $R = \sqrt{R_x^2 + R_z^2} = \sqrt{101.2^2 + 246.1^2} \quad = 266 \text{ N}$

The angle of $\vec{R}$ from the $x$ direction is: $\alpha = \tan^{-1}\dfrac{R_z}{R_x} = \tan^{-1}\dfrac{246.1}{101.2} = 67.6°$

### 3.3 Derivation of the Moment-of-Momentum Equation

The moment-of-momentum equation is crucial in many engineering problems, particularly when addressing torque-related issues. It is based on Newton's second law, which links forces to linear momentum flow, allowing for the consideration of torques as well. By analyzing the moment of linear momentum and the resultant forces acting on fluid particles, we can develop the moment-of-momentum equation. This equation is often more useful than the linear momentum equation when dealing with torques and angular momentum flow.

Application of Newton's second law of motion to a particle of fluid (Figure 2.7) yields:

$$\frac{D}{Dt}(\vec{V}\rho\delta\vartheta) = \delta\vec{F}_{particle}$$

If we form the moment of each side with respect to the origin of an inertial coordinate system, we obtain:

$$\vec{\mathbf{r}} \times \frac{D}{Dt}(\vec{V}\rho\delta\vartheta) = \vec{\mathbf{r}} \times \delta\vec{F}_{particle}$$

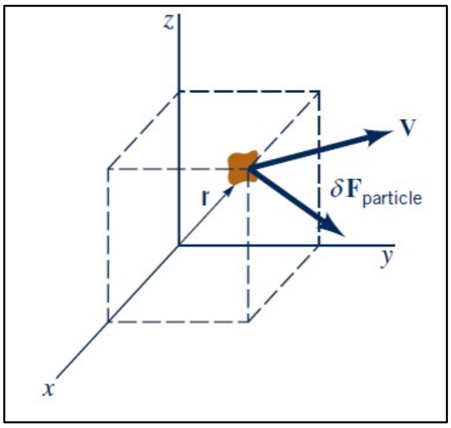



**Figure 2.7** Inertial coordinate system

Note that $\dfrac{D}{Dt}\left[(\vec{\mathbf{r}}\times\vec{V})\rho\delta\vartheta\right]=\dfrac{D\vec{\mathbf{r}}}{Dt}\times\vec{V}\rho\delta\vartheta+\vec{\mathbf{r}}\times\dfrac{D(\vec{V}\rho\delta\vartheta)}{Dt}$

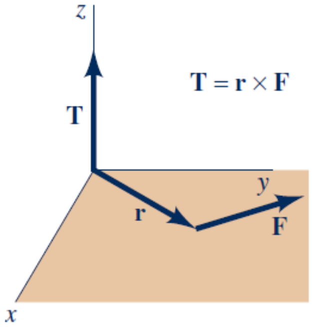
Knowing that $\dfrac{D\vec{\mathbf{r}}}{Dt}=\vec{V}$ **and** $\vec{V}\times\vec{V}=\mathbf{0}$

We obtain,

$$\frac{D}{Dt}\left[(\vec{\mathbf{r}}\times\vec{V})\rho\delta\vartheta\right]=\vec{\mathbf{r}}\times\delta\vec{F}_{particle}$$

For a system (collection of fluid particles), we need to use the sum of both sides to obtain:

$$\int_{sys}\frac{D}{Dt}\left[(\vec{\mathbf{r}}\times\vec{V})\rho d\vartheta\right]=\sum\vec{\mathbf{r}}\times d\vec{F}_{particle}$$




where $\sum\vec{\mathbf{r}}\times\delta\vec{F}_{particle}=\sum(\vec{\mathbf{r}}\times\vec{F})_{sys}$

Note that $\displaystyle\int_{sys}\frac{D}{Dt}\left[(\vec{\mathbf{r}}\times\vec{V})\rho d\vartheta\right]=\frac{D}{Dt}\int_{sys}(\vec{\mathbf{r}}\times\vec{V})\rho d\vartheta$

Hence,

$$\int_{sys}\frac{D}{Dt}\left[(\vec{\mathbf{r}}\times\vec{V})\rho d\vartheta\right]=\sum(\vec{\mathbf{r}}\times\vec{F})_{sys}$$

| Time rate of change of the moment-of-momentum of the system | = sum of external torques acting on the system |
|---|---|

For a control volume that is instantaneously coincident with the system, the torques acting on the system and on the control volume contents will be identical:

$$\sum(\vec{\mathbf{r}}\times\vec{F})_{sys}=\sum(\vec{\mathbf{r}}\times\vec{F})_{CV}$$

Further, for the system and the contents of the coincident control volume that is fixed and non-deforming, the Reynolds transport theorem leads to:

$$\frac{D}{Dt}\int_{sys}(\vec{\mathbf{r}}\times\vec{V})\rho d\vartheta=\frac{\partial}{\partial t}\int_{CV}(\vec{\mathbf{r}}\times\vec{V})\rho d\vartheta+\int_{CS}(\vec{\mathbf{r}}\times\vec{V})\rho\vec{V}.\vec{n}dA$$

Hence,

$$\boxed{\frac{\partial}{\partial t}\int_{CV}(\vec{\mathbf{r}}\times\vec{V})\rho d\vartheta+\int_{CS}(\vec{\mathbf{r}}\times\vec{V})\rho\vec{V}.\vec{n}dA=\sum(\vec{\mathbf{r}}\times\vec{F})_{CV}} \quad (2.11)$$

### 3.4 Application of the Moment-of-Momentum Equation

**Example 3.14** **Torque**

Water enters a rotating lawn sprinkler through its base at the steady rate of $1000\ ml/s$ as sketched in Figure E3.14A. The exit area of each of the two nozzles is 30 mm2 and the flow leaving each nozzle is in the tangential direction. The radius from the axis of rotation to the centerline of each nozzle is 200 mm.

(a) Determine the resisting torque required to hold the sprinkler head stationary.

(b) Determine the resisting torque associated with the sprinkler rotating with a constant speed of 500 rev/min.

(c) Determine the speed of the sprinkler if no resisting torque is applied.

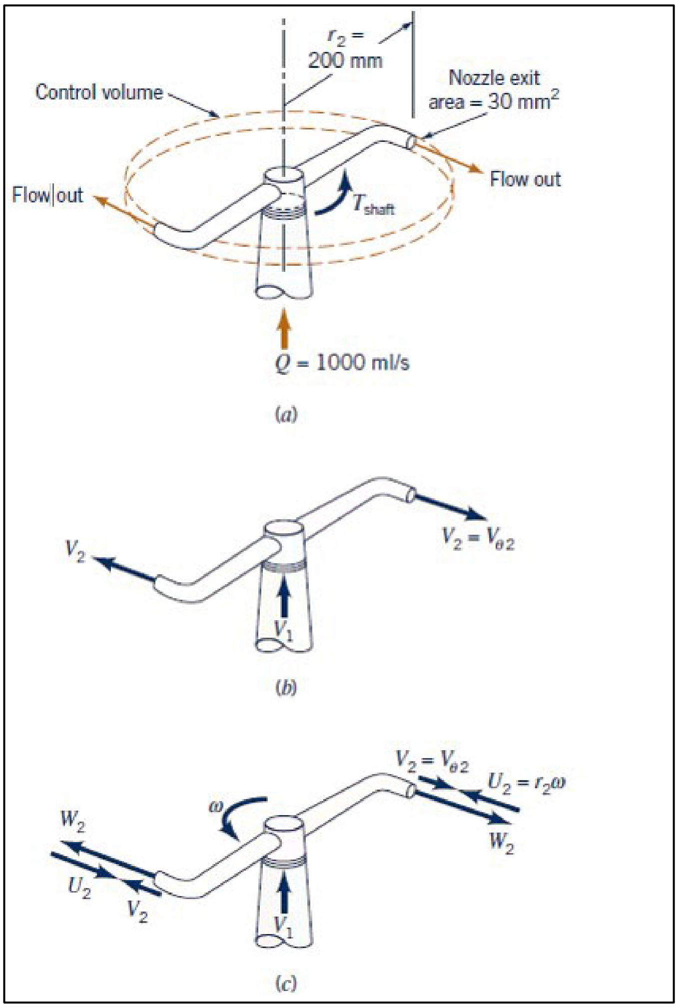


**Figure E 3.14A**

Before starting to solve this example, it is useful to make a few assumptions in order to simplify Eq. 3.11 and thus obtain certain results, which follow from it precisely to solve this exercise.

We assume that flows considered are one-dimensional (uniform distributions of average velocity at any section).

We confine ourselves to steady or steady-in-the-mean cyclical flows. Thus,

$$\frac{\partial}{\partial t}\int_{CV}(\vec{\mathbf{r}}\times\vec{V})\rho d\vartheta = 0$$

We work only with the component of moment-of-momentum equation resolved along the axis of rotation.

Figure E 3.14B help to better understand the subsequent analysis.

The absolute and relative velocities, $\vec{V}$ and $\overrightarrow{W}$, are related by the vector relationship:

$$\vec{V} = \vec{W} + \vec{U}$$

where $U$ is the velocity of the moving nozzle as measured relative to the fixed control surface.

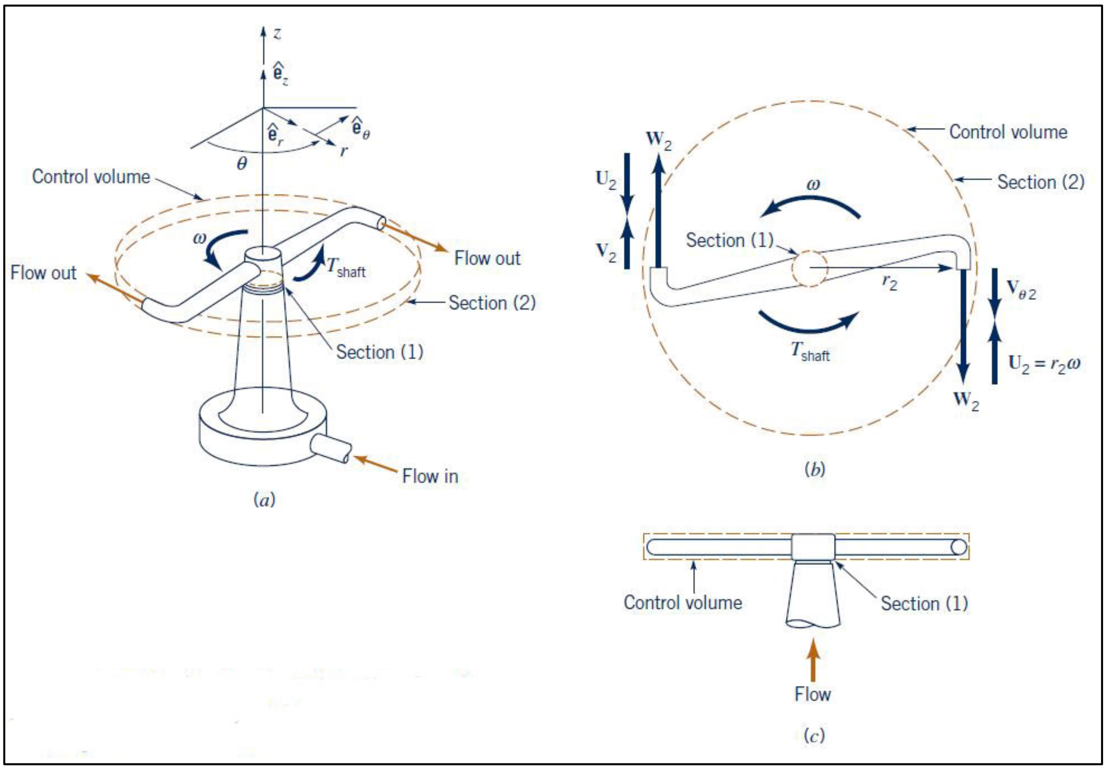



**Figure E 3.14B**

The second term on the left of Eq. 3.11 can be written as follows:

$$\int_{CS} (\vec{\mathrm{r}} \times \vec{V})\rho\vec{V}.\vec{n}\delta A = (-r_2 V_{\theta 2})(+\dot{m}_2)$$

The term on the right-hand side of Eq. 3.11 can be expressed as:

$$\left[\sum (\vec{\mathrm{r}} \times \vec{F})_{\substack{\text{contents of the}\\ \text{control volume}}}\right] = T_{shaft}$$

$T_{shaft}$ represents the net torque acting with respect to the axis of rotation.

Hence,

$$-r_2 V_{\theta 2}\dot{m}_2 = T_{shaft}$$

The shaft power is expressed as follows:

$$\dot{W}_{shaft} = T_{shaft}\omega = -r_2 V_{\theta 2}\dot{m}_2\omega$$

With $r_2\omega_2 = U$, we can write:

$$\dot{W}_{shaft} = -U V_{\theta 2}\dot{m}_2$$

The shaft work per unit mass is then:

$w_{shaft} = (\dot{W}_{shaft} / \dot{m}_2) = -UV_{\theta 2}$

**Now let's go back to our example**,

$T_{shaft} = -r_2 V_{\theta 2} \dot{m}_2$

Since the control volume is fixed and non-deforming and the flow exiting from each nozzle is tangential, $V_{\theta 2} = V_2 = 16.7$ m/s (See example 3.8, continuity equation)

$\Rightarrow \quad T_{shaft} = -r_2 V_2 \dot{m}_2$

$\dot{m}_2 = \rho Q = 1000 \times 1000 \times 10^{-6} = 1$ kg/s

$\Rightarrow \quad T_{shaft} = -200 \times 10^{-3} \times 16.7 \times 1 = -3.34$ N.m

When the sprinkler is rotating at a constant speed of $500\ rpm$, the absolute velocity of the fluid leaving each nozzle, $V_2$, is :

$V_2 = W_2 - U_2$

where,

$W_2 = 16.7$ m/s and
$U_2 = r_2\omega = 200 \times 10^{-3} \times 500 \times 2\pi / 60 \simeq 10.5$ m/s

$\Rightarrow \quad V_2 = 16.7 - 10.5 = 6.2$ m/s

$\Rightarrow \quad T_{shaft} = -200 \times 10^{-3} \times 6.2 \times 1 = -1.24$ N.m

When no resisting torque is applied to the rotating sprinkler head, a maximum constant speed of rotation will occur as demonstrated below.

$T_{shaft} = -r_2 (W_2 - r_2 \omega_2) \dot{m}_2$

For no resisting torque, $0 = -r_2 (W_2 - r_2 \omega) \dot{m}_2$

$\omega = W_2 / r_2$

$\omega = 16.7 / 0.2 = 83.5$ rad/s $\simeq 798$ rpm

*For this condition ($T_{shaft} = 0$), the water both enters and leaves the control volume with zero angular momentum.*

When the moment-of-momentum equation is applied to a more general, one dimensional flow through a rotating machine, we obtain:

$$T_{shaft} = (-\dot{m}_{in})(\pm r_{in} V_{\theta in}) + \dot{m}_{out}(\pm r_{out} V_{\theta out})$$

The " – " is used with mass flowrate into the control volume $\dot{m}_{in}$, and the " + " is used with mass flowrate out of the control volume $\dot{m}_{out}$, to account for the sign of the dot product, $\vec{V} \times \vec{n}$.

A simple way to determine the sign of the product $rV_\theta$ is to compare the direction of $V_\theta$ and $U$. If $V_\theta$ and $U$ are in the same direction, then the product is positive. If $V_\theta$ and $U$ are in opposite directions, the product is negative. The sign of the shaft torque is " + " if $T_{shaft}$ is in the same direction along the axis of rotation as $\omega$, and otherwise " – ".

The shaft power, $W_{shaft}$, is related to shaft torque, $T_{shaft}$, by:

$$\dot{W}_{shaft} = \dot{T}_{shaft}\omega$$

$$\Rightarrow \quad \dot{W}_{shaft} = (-\dot{m}_{in})(\pm r_{in}\omega V_{\theta in}) + \dot{m}_{out}(\pm r_{out}\omega V_{\theta out})$$

or, since $r\omega = U$,

$$\dot{W}_{shaft} = (-\dot{m}_{in})(\pm U_{in} V_{\theta in}) + \dot{m}_{out}(\pm U_{out} V_{\theta out})$$

Conservation of mass: $\dot{m}_{in} = \dot{m}_{out} = \dot{m}$

$$\Rightarrow \quad w_{shaft} = -(\pm U_{in} V_{\theta in}) + (\pm U_{out} V_{\theta out})$$

## 4. CHAPTER SUMMARY

This chapter introduces the foundational concepts of finite control volume analysis, which is crucial for understanding fluid dynamics. The core approach involves the Reynolds Transport Theorem (RTT) and its application to mass, momentum, and energy conservation in control volumes. The chapter focuses on the interaction between fluid systems and their surroundings, analyzing how forces, moments, and fluid properties evolve within a fixed, moving, or deforming control volume.

The primary goal is to apply the Reynolds Transport Theorem to connect system analysis with control volume analysis, particularly for steady and unsteady, compressible and incompressible flows. This is achieved by examining mass flow, momentum exchanges, and angular momentum in a wide range of practical applications, including jet engines, hydraulic systems, and rotating machinery.

Key sections discuss:

***System vs. Control Volume***: Differentiates between a system (fixed matter) and a control volume (spatial region where fluid flows), highlighting the importance of boundary conditions.

***Reynolds Transport Theorem***: Derived and explained as a method to link the change of properties in a system with those in a control volume, with applications to mass, velocity, temperature, and momentum.

***Conservation of Mass (Continuity Equation)***: Describes the principle of mass conservation through the continuity equation, ensuring that mass entering a control volume equals the mass leaving it, except for any change within the volume.

***Newton's Second Law***: The linear momentum equation, derived using Newton's second law, relates the forces acting on a system to the change in momentum. The chapter explores applications of this equation in fluid systems, including jet flows and vane analysis.

***Moment-of-Momentum Equation***: Discusses how torque and angular momentum are analyzed in rotating systems, providing crucial insights for designing machinery like turbines and lawn sprinklers.

The chapter concludes by emphasizing the practical applications of these principles in engineering, highlighting their importance in the design and analysis of complex fluid systems. Through numerous examples, the chapter illustrates how these theoretical foundations can be used to solve real-world engineering problems.

# DIMENSIONAL ANALYSIS AND SIMILITUDE

## 1. INTRODUCTION

Dimensional analysis, similitude, and modeling are fundamental techniques in fluid mechanics and engineering, enabling the systematic study of physical phenomena through the reduction of complex systems to simpler, dimensionless forms. The primary objective of these methods is to allow engineers to derive generalized solutions for a wide range of practical problems, especially when direct analytical solutions are unattainable due to the complexities of real-world systems.

In many engineering applications, especially in fluid dynamics, it is often impractical to rely solely on theoretical and numerical analyses. Instead, experimental data plays a crucial role in developing accurate models and making reliable predictions. By using dimensional analysis, engineers can simplify the experimental process by reducing the number of variables involved and focusing on dimensionless groups that capture the essential behavior of the system under study.

Similitude, closely related to dimensional analysis, allows for the use of laboratory models to simulate real-world systems. The ability to apply experimental findings from small-scale models to full-scale systems is essential in engineering, as it allows for cost-effective and time-efficient testing under controlled conditions. Through the use of dimensionless parameters, engineers can relate the behavior of a model to its real-world counterpart, ensuring that predictions made from experimental data are applicable beyond the laboratory setting.

In this chapter, we will explore the techniques of dimensional analysis and similitude in fluid mechanics. We will cover the Buckingham $\pi$ theorem, which is central to forming dimensionless groups, and learn how these groups can be used to derive prediction equations. Additionally, the chapter will provide insight into the systematic application of these methods in experimental design and data interpretation, helping engineers draw meaningful conclusions from complex, real-world fluid systems.

## 2. DIMENSIONAL ANALYSIS

***Definition***

Dimensional analysis is a method that simplifies complex problems in fluid mechanics by reducing the number of variables through the formation of dimensionless groups (also known as pi terms). These groups are combinations of variables such that they have no dimensions. The basic dimensions typically used are mass ($M$), length ($L$), and time ($T$), or sometimes force ($F$), length ($L$), and time ($T$), which are related by Newton's second law.

***Why Use Dimensional Analysis?***

The dimensional analysis is used to:

Simplifies complex systems by reducing the number of variables.

Helps generalize experimental results from models to prototypes.

Identifies key variables that significantly influence the system, aiding in predictions.

***Dimensional Homogeneity***

A core principle in dimensional analysis is dimensional homogeneity. This principle states that for any physical equation to be valid, the dimensions of all terms must be consistent. Each term on both sides of the equation must have the same dimensions. For example, consider the equation for kinetic energy:

$$E_k = \frac{1}{2} m V^2$$

where:

$E_k \equiv$ Kinetic energy (dimension: $[M\ L^2\ T^{-2}]$)

$m \equiv$ Mass (dimension: $[M]$)

$V \equiv$ velocity (dimension: $[L\ T^{-1}]$)

If we check the dimensions on both sides:

Left side: $[M\ L^2\ T^{-2}]$

Right side: $[M] \times [L\ T^{-1}]^2 = [M\ L^2\ T^{-2}]$

Since both sides match, the equation is dimensionally homogeneous.

***The Buckingham Pi Theorem***

The Buckingham Pi Theorem states the following:

*"If an equation involving $\boldsymbol{k}$ variables is dimensionally homogeneous, it can be reduced to a relationship among $\boldsymbol{k-r}$ independent dimensionless products, where $\boldsymbol{r}$ is the minimum number of reference dimensions required to describe the variables".*

The dimensionless products are frequently referred to as ***pi terms***.

The pi theorem is based on the idea of dimensional homogeneity. Essentially, we assume that for any physically meaningful equation involving $\boldsymbol{k}$ variables, such as:

$x_1 = f(x_2, x_3, \ldots, x_k)$

The dimensions of the variable on the left side of the equal sign must be equal to the dimensions of any term that stands by itself on the right side of the equal sign. It then follows that we can rearrange the equation into a set of dimensionless products (pi terms) so that:

$$\Pi_1 = g(\Pi_2,\ \Pi_3,\ \ldots,\ \Pi_{k-r})$$

where $g(\Pi_2,\ \Pi_3,\ \ldots,\ \Pi_{k-r})$ is a function of $\Pi_2$ through $\Pi_{k-r}$.

***Determination of Pi Terms***

Several methods can be used to form the dimensionless products, or pi terms. The method we will describe in detail in this section is called the ***method of repeating variables***.

**Steps to Apply the Method**:

List all the variables that are involved in the problem.

Express each of the variables in terms of basic dimensions.

Determine the required number of pi terms.

Select a number of repeating variables, where the number required is equal to the number of reference dimensions.

Form a pi term by multiplying one of the nonrepeating variables by the product of the repeating variables, each raised to an exponent that will make the combination dimensionless.

Repeat Step 5 for each of the remaining nonrepeating variables.

Check all the resulting pi terms to make sure they are dimensionless.

Express the final form as a relationship among the pi terms, and think about what it means.

To illustrate these various steps we consider the steady flow of an incompressible Newtonian fluid through a long, smooth-walled, horizontal, circular pipe (Figure 3.1). For this flow, it can be demonstrated that the pressure difference, per unit length, $\Delta p_\ell$ depends on certain factors or variables:

$$\Delta p_\ell = f(D, \rho, \mu, V)$$

where, $D$ is the pipe diameter, $\rho$ is the fluid density, $\mu$ is the fluid viscosity, and $V$ represents the mean velocity at which the fluid is flowing through the pipe.

Setting up this list of variables is the ***first step***.

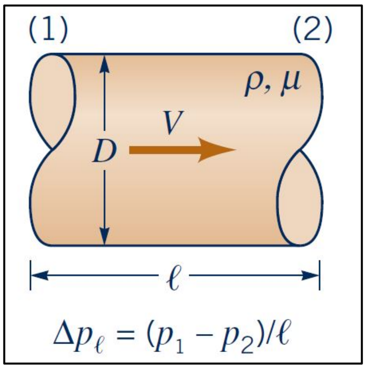



**Figure 3.1** Pressure difference per unit length in a pipe

Next (***Step 2***) we express all the variables in terms of basic dimensions. Using $F$, $L$, and $T$ as basic dimensions (we could also use $M$, $L$, and $T$ as basic dimensions if desired- the final result will be the same.), it follows that:

$$\Delta p_\ell \doteq (FL^{-2})L^{-1} \doteq FL^{-3}$$
$$D \doteq L$$
$$\rho \doteq FL^{-4}T^{2}$$
$$\mu \doteq FL^{-2}T$$
$$V \doteq LT^{-1}$$

Note the sign $\doteq$ is used to indicate dimensional equality.

We can now apply the pi theorem to determine the required number of pi terms (***Step 3***). We see that there are five variables ($k = 5$) and all three basic dimensions are required to describe the variables ($r = 3$). Consequently, according to the pi theorem there will be $(5 - 3)$ or two pi terms required.

The repeating variables to be used to form the pi terms (***Step 4***) need to be selected from the list $D, \rho, \mu$ and $V$. In this example we will use $D, \rho$ and $V$ as repeating variables. Note that these are dimensionally independent, since $D$ is a length, $V$ involves both length and time, and $\rho$ involves force, length, and time. This means that we cannot form a dimensionless product from this set.

We are now ready to form the two pi terms (***Step 5***). Typically, we would start with the dependent variable and combine it with the repeating variables to form the first pi term; that is,

$$\Pi_1 = \Delta p_\ell D^a V^b \rho^c$$

Since this combination is to be dimensionless, it follows that,

$$(FL^{-3})(L)^a(LT^{-1})^b(FL^{-4}T^2)^c \doteq F^0L^0T^0$$

By comparing the exponents of each dimension on both sides of this equality, we obtain the following system:

$$\begin{cases} 1 + c = 0 \\ -3 + a + b - 4c = 0 \\ -b + 2c = 0 \end{cases}$$

It follows that $\boldsymbol{a = 1, b = -2, c = -1,}$ and therefore,

$$\Pi_1 = \frac{\Delta p_\ell D}{\rho V^2}$$

The process is now repeated for the remaining nonrepeating variables (***Step 6***). In this example there is only one additional variable ($\mu$) so that:

$$\Pi_2 = \mu D^a V^b \rho^c$$

Following the same method as in step 5, we find $\boldsymbol{a = -1, b = -1, c = -1,}$ and therefore,

$$\Pi_2 = \frac{\mu}{DV\rho}$$

At this point stop and check to make sure the pi terms are actually dimensionless (***Step 7***). We will check using both $FLT$ and $MLT$ dimensions. Thus,

$$\Pi_1 = \frac{\Delta p_\ell D}{\rho V^2} \doteq \frac{(FL^{-3})(L)}{(FL^{-4}T^2)(LT^{-1})^2} \doteq F^0 L^0 T^0$$

$$\Pi_2 = \frac{\mu}{DV\rho} \doteq \frac{(FL^{-2}T)}{(L)(LT^{-1})(FL^{-4}T^2)} \doteq F^0 L^0 T^0$$

or alternatively,

$$\Pi_1 = \frac{\Delta p_\ell D}{\rho V^2} \doteq \frac{(ML^{-2}T^{-2})(L)}{(ML^{-3})(LT^{-1})^2} \doteq M^0 L^0 T^0$$

$$\Pi_2 = \frac{\mu}{DV\rho} \doteq \frac{(ML^{-1}T^{-1})}{(L)(LT^{-1})(ML^{-3})} \doteq M^0 L^0 T^0$$

Finally (***Step 8***), we can express the result of the dimensional analysis as:

$$\frac{\Delta p_\ell D}{\rho V^2} = g\left(\frac{\mu}{DV\rho}\right)$$

Dimensional analysis will not provide the form of the function $\mathbf{g}$. This can only be obtained from a suitable set of experiments. If desired, the pi terms can be rearranged; that is, the reciprocal of $\mu/DV\rho$ could be used and, of course, the order in which we write the variables can be changed. Thus, for example, $\Pi_2$ could be expressed as:

$$\Pi_2 = \frac{\rho VD}{\mu}$$

and the relationship between $\Pi_1$ and $\Pi_2$ as:

$$\frac{D\Delta p_\ell}{\rho V^2} = G\left(\frac{\rho VD}{\mu}\right)$$

The dimensionless product $\rho VD/\mu$ is a very famous one in fluid mechanics; it is called the ***Reynolds number***. The dimensionless product ρVD/μ is a very famous one in fluid

**Example 4.1** **Three pi terms**

We consider the steady incompressible flow around a cylinder of diameter $D$ and length $\ell$. The drag force $F_D$ on the cylinder is to be studied. What functional form relates the dimensionless variables if a fluid with velocity $V$ flows normal to the cylinder?

**Solution**

In this example we will include as influential variables the free streamvelocity $V$, the viscosity $\mu$, the density $\rho$ of the fluid, in addition to the diameter $D$ and the length $\ell$ of the cylinder, resulting in $k = 6$ variables. This is written as:

$$F_D = f(D, \ell, \rho, \mu, V)$$

The variables are observed to include all the three basic dimensions $MLT$ ( $r = 3$):

$$F_D \doteq MLT^{-2}$$
$$D \doteq L$$
$$\ell \doteq L$$
$$\rho \doteq ML^{-3}$$
$$\mu \doteq ML^{-1}T^{-1}$$
$$V \doteq LT^{-1}$$

Consequently, we can expect $\boldsymbol{k - r = 6 - 3 = 3}$ pi terms.

We choose repeating variables with the simplest combinations of dimensions such that they do not form a pi term by themselves (we could not include $D$ and $\ell$ as repeating variables); the repeating variables are chosen to be$D$, $V$, and $\rho$.

We now move on to form the three pi terms:

$$\Pi_1 = F_D D^a V^b \rho^c$$

$$(MLT^{-2})(L)^a (LT^{-1})^b (ML^{-3})^c \doteq M^0 L^0 T^0$$

$$\Rightarrow \quad \begin{cases} 1+c=0 \\ 1+a+b-3c=0 \\ -2-b=0 \end{cases}$$

$$\Rightarrow \quad a=-2, b=-2, c=-1$$

$$\Rightarrow \quad \Pi_1 = F_D D^{-2} V^{-2} \rho^{-1} = \frac{F_D}{\rho V^2 D^2}$$

$$\Pi_2 = \ell D^a V^b \rho^c$$

$$(L)(L)^a (LT^{-1})^b (ML^{-3})^c \doteq M^0 L^0 T^0$$

$$\Rightarrow \quad \begin{cases} 3c=0 \\ 1+a+b-3c=0 \\ -b=0 \end{cases}$$

$$\Rightarrow \quad a=-1, b=0, c=0$$

$$\Pi_2 = \frac{\ell}{D}$$

$$\Pi_3 = \mu D^a V^b \rho^c$$

$$(ML^{-1}T^{-1})(L)^a (LT^{-1})^b (ML^{-3})^c \doteq M^0 L^0 T^0$$

$$\Rightarrow \quad \begin{cases} 1+c=0 \\ -1+a+b-3c=0 \\ -1-b=0 \end{cases}$$

$$\Rightarrow \quad a=-1, b=-1, c=-1$$

$$\Pi_3 = \frac{\mu}{\rho V D}$$

The dimensionless, functional relationship relating the pi terms is:

$$\Pi_1 = g(\Pi_2, \Pi_3) \quad \text{or} \quad \frac{F_D}{\rho V^2 D^2} = g\left(\frac{\ell}{d}, \frac{\mu}{\rho V D}\right)$$

**<u>Example 4.2</u>** **One pi term**

At low velocities (laminar flow), the volume flow $Q$ through a small-bore tube is a function only of the tube radius $R$, the fluid viscosity $\mu$, and the pressure drop per unit tube length $dp/dx$. Using the pi theorem, find an appropriate dimensionless relationship.

**Solution**

Write the given relation and count variables:

$Q = f(R,\ \mu,\ dp/dx)$ four variables ($k = 4$)

$$
\begin{aligned}
&Q \doteq L^3T^{-1}\\
&R \doteq L\\
&\mu \doteq ML^{-1}T^{-1}\\
&dp/dx \doteq ML^{-2}T^{-2}
\end{aligned}
$$

The three basic dimensions are required ($r = 3$). Hence, one pi term is to be determined.

$$\Pi_1 = QR^a\mu^b(dp/dx)^c$$

$$\Rightarrow \quad (L^3T^{-1})(L)^a(ML^{-1}T^{-1})^b(ML^{-2}T^{-2})^c \doteq M^0L^0T^0$$

$$\Rightarrow \quad \begin{cases} b+c=0 \\ 3+a-b-2c=0 \\ -1-b-2c=0 \end{cases}$$

Solving simultaneously, we obtain $a = -4$, $1$, $c = -1$. Then

$$\Pi_1 = QR^{-4}\mu^1(dp/dx)^{-1} = \frac{Q\mu}{R^4(dp/dx)}$$

$$\Rightarrow \quad \Pi_1 = g(\text{nothing}) = \text{constant} \quad \text{or} \quad \frac{Q\mu}{R^4(dp/dx)} = \text{constant}$$

***Common Dimensionless Groups in Fluid Mechanics***

Several dimensionless groups are commonly used to simplify problems in fluid mechanics and help identify dominant forces. Key Dimensionless groups are listed below:

***Reynolds Number (Re)***

$$Re = \frac{\rho VL}{\mu}$$

Reynolds number represents the ratio of inertial forces to viscous forces. It is fundamental in determining whether a flow is laminar or turbulent.

***Froude Number (Fr)***

$$Fr = \frac{V}{\sqrt{gL}}$$

The Froude number represents the ratio of inertial forces to gravitational forces and is especially important in flows with free surfaces such as rivers or around ships.

***Euler Number (Eu)***

$$Eu = \frac{\Delta p}{\rho V^2}$$

The Euler number is the ratio of pressure forces to inertial forces. It often appears in the form of a pressure coefficient and is used in problems where pressure differences play a significant role.

***Cauchy Number (Ca)* and *Mach Number (Ma)***

$$Ca = \frac{\rho V^2}{K}, \quad Ma = \frac{V}{c}$$

$K$ is the bulk modulus and $c$ is the speed of sound.

The Cauchy number and the Mach number both measure the ratio of inertial to compressibility effects. Thus, either number (but not both) may be used in problems in which fluid compressibility is important.

***Strouhal Number (St)***

$$St = \frac{\omega L}{V}$$

Where, $\omega$ represents the characteristic frequency of the unsteady or periodic phenomenon in the flow.

The Strouhal number represents the ratio of inertial forces due to unsteady (local acceleration) effects to those due to convective acceleration. It is relevant in oscillating flows and vortex shedding phenomena, such as flow past cylinders.

***Weber Number*** **(*We*)**

$$We = \frac{\rho V^2 L}{\sigma}$$

Where,$\sigma$ represents the surface tension of the fluid.

The Weber number compares inertial forces to surface tension forces. It is significant in multiphase flows involving interfaces between fluids, such as droplets or bubbles.

## 3. SIMILITUDE

The core principle of modeling and similitude establishes that physical models, which are scaled representations of prototypes or actual systems, can predict prototype behavior when designed according to similarity laws. This approach relies on the Buckingham Pi Theorem, which states that a dimensionally homogeneous equation with $k$ variables reducible to $k - r$ independent dimensionless groups (Pi terms) enables similarity between systems.

For a prototype governed by:

$$\Pi_1 = f(\Pi_2, \Pi_3, ..., \Pi_n)$$

a similar relationship can be written for a model of this prototype; that is,

$$\Pi_{1m} = f(\Pi_{2m}, \Pi_{3m}, ..., \Pi_{nm})$$

Similarity requirements demand equality of all independent Pi terms:

$$\Pi_{2m} = \Pi_2, \Pi_{3m} = \Pi_3, ..., \Pi_{nm} = \Pi_n$$

This ensures the dependent Pi term ($\Pi_1$) remains identical between model and prototype ($\Pi_1 = \Pi_{1\mathrm{m}}$).

The model design process involves:

***Geometric similarity***: *A model and prototype are geometrically similar if and only if all body dimensions in all three coordinates have the same linear-scale ratio* ($\frac{l_m}{l_p} = \alpha$).

All angles are preserved in geometric similarity. All flow directions are preserved. The orientations of model and prototype with respect to the surroundings must be identical (Figure 3.2).

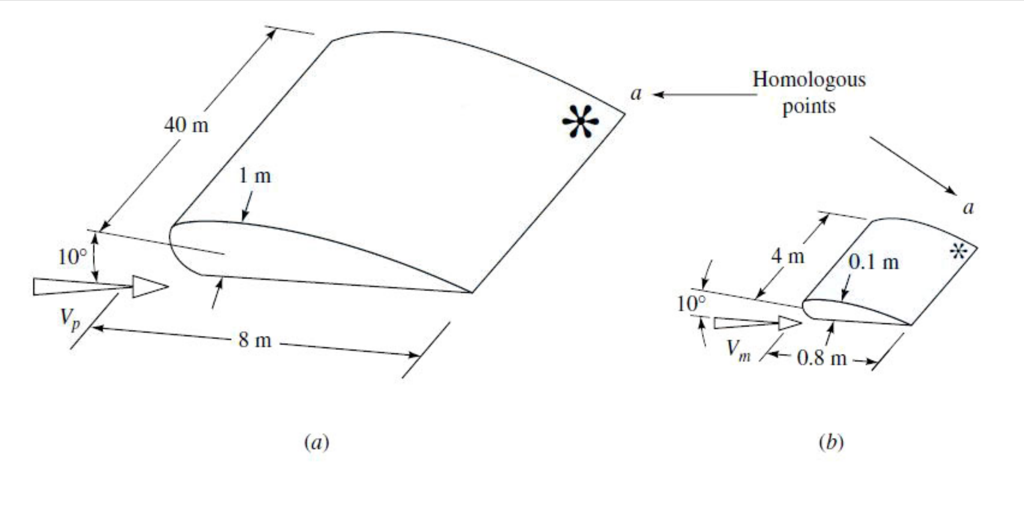



**Figure 3.2** Geometric similarity: (a) prototype; (b) one-tenth-scale model

***Kinematic Similarity***: Kinematic similarity requires that the model and prototype have the same length-scale ratio ($\frac{l_m}{l_p} = \alpha$) and the same time-scale ratio ($\frac{T_m}{T_p} = \tau$). The result is that the velocity-scale ratio ($\frac{V_m}{V_p} = \beta$) will be the same for both.

*The motions of two systems are kinematically similar if homologous particles lie at homologous points at homologous times.*

For flows without free surfaces, kinematic similarity can be achieved without needing additional parameters. For systems with free surfaces, such as waves, the Froude number becomes important to maintain kinematic similarity (see Figure 3.3a and b).

$$Fr_m = Fr_p \quad \Rightarrow \quad \frac{V_m}{\sqrt{gL_m}} = \frac{V_p}{\sqrt{gL_p}}$$

$$\Rightarrow \quad \frac{V_m}{V_p} = \sqrt{\frac{L_m}{L_p}} = \sqrt{\alpha}$$

$$\Rightarrow \quad \frac{T_m}{T_p} = \frac{L_m / V_m}{L_p / V_p} = \sqrt{\alpha}$$

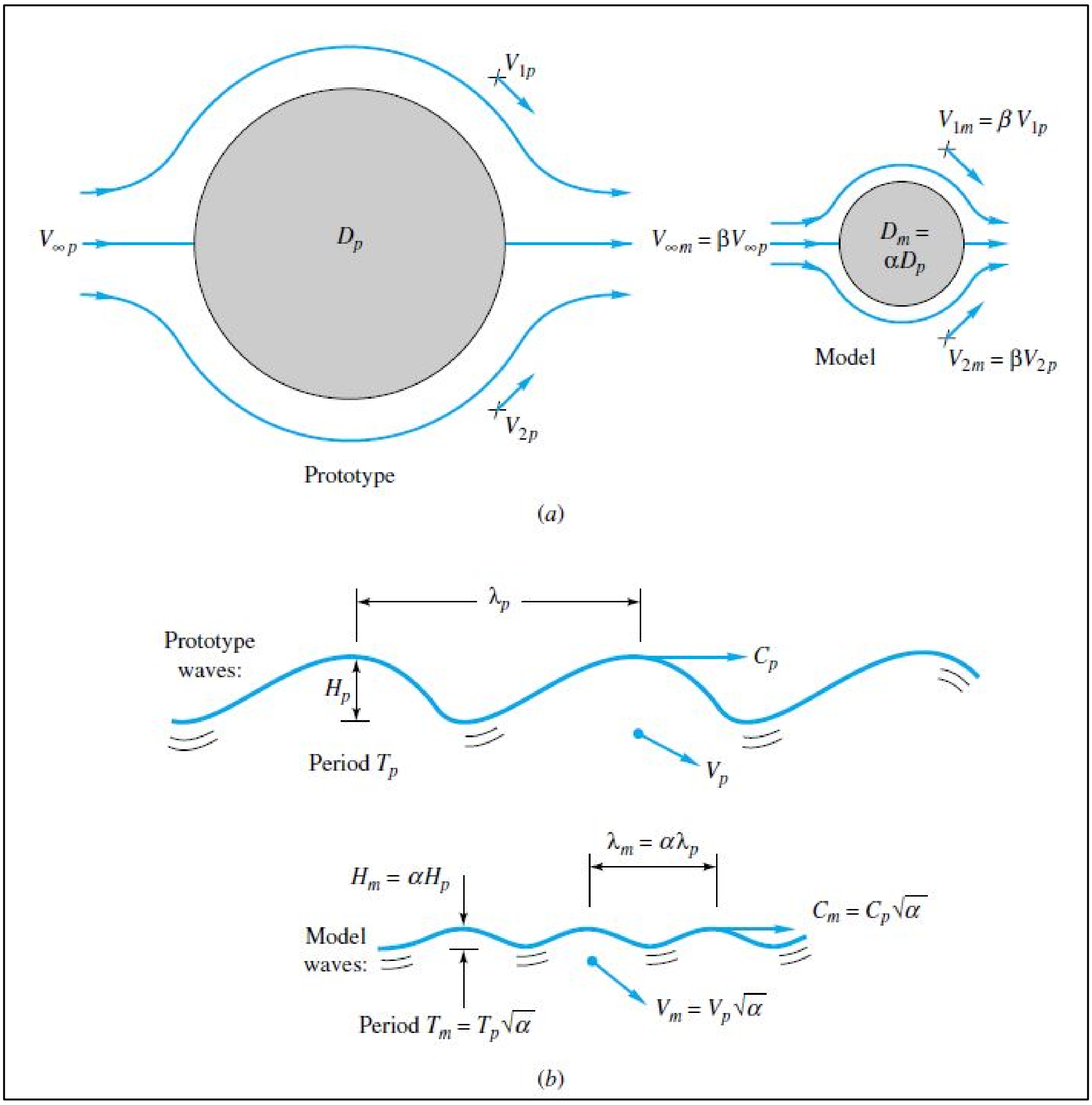



**Figure 3.3** Kinematic similarity (a) Frictionless low-speed flows, (b) free-surface flow

***Dynamic Similarity***: *Dynamic similarity occurs when both the model and prototype have the same length-scale, time-scale, and force-scale ratios.*

In other words, dynamic similarity exists, simultaneous with kinematic similarity, if the model and prototype force and pressure coefficients are identical. This is ensured if:

For compressible flow, the model and prototype Reynolds number and Mach number and specific-heat ratio are correspondingly equal.

For incompressible flow,

With no free surface: model and prototype Reynolds numbers are equal.

With a free surface: model and prototype Reynolds number, Froude number, and (if necessary) Weber number and Euler number are correspondingly equal.

Mathematically, Newton's law for any fluid particle requires that the sum of the pressure force, gravity force, and friction force equal the acceleration term, or inertia force:

$$\vec{F}_p + \vec{F}_g + \vec{F}_f = \vec{F}_i$$

The dynamic-similarity laws listed above ensure that each of these forces will be in the same ratio and have equivalent directions between model and prototype

Figure 3.4 shows an example for flow through a sluice gate. The force polygons at homologous points have exactly the same shape if the Reynolds and Froude numbers are equal (neglecting surface tension and cavitation, of course).

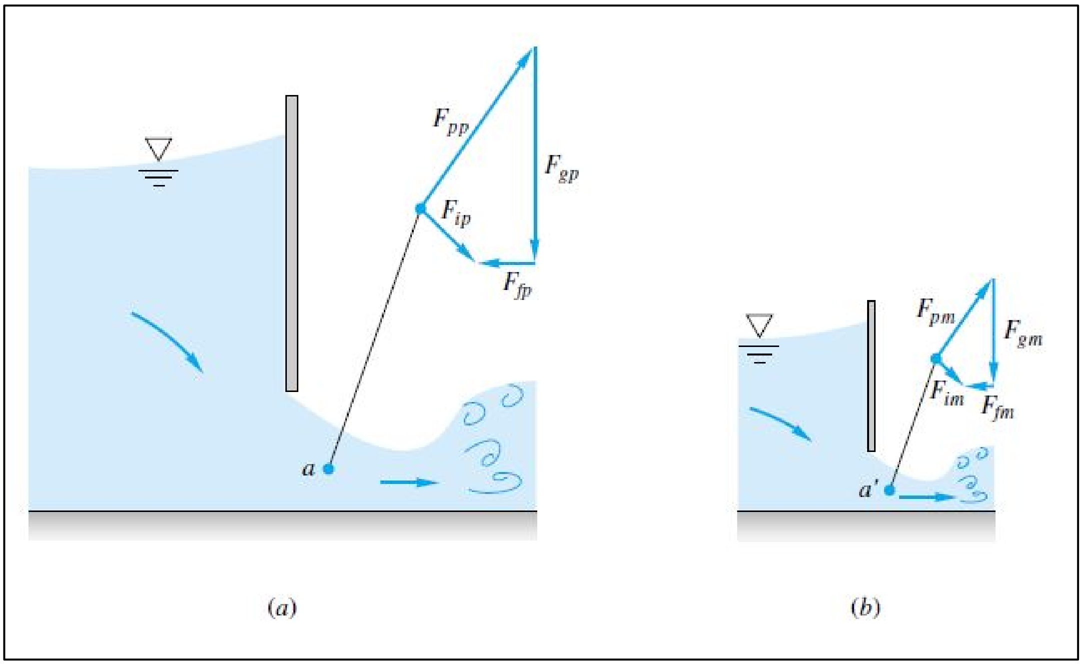



**Figure 3.4** Dynamic similarity illustration (a) Prototype, (b) model

## 4. APPLICATIONS

**Example 4.3**

A thin rectangular plate having a width $w$ and a height $h$ is located so that it is normal to a moving stream of fluid as shown in Figure E4.2. Assume the drag $\mathfrak{D}$ that the fluid exerts on the plate is a function of $w$ and $h$, the fluid viscosity and density, $\mu$ and $\rho$, respectively, and the velocity $V$ of the fluid approaching the plate.

Determine a suitable set of pi terms to study this problem experimentally.

Find a relationship between the drag force on a certain prototype and the drag on a model so that the similarity requirements are satisfied.

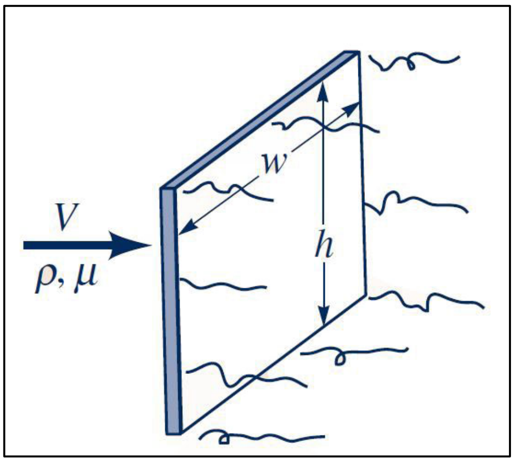



**Figure E 4.3**

From the statement of the problem, we can write:

$\mathcal{D} = f(w,\ h,\ \mu,\ \rho,\ V)$, six variables ($k = 6$)

The dimensions of the variables (using the MLT system) are:

$$\mathcal{D} \doteq MLT^{-2}$$
$$w \doteq L$$
$$h \doteq L$$
$$\mu \doteq ML^{-1}T^{-1}$$
$$\rho \doteq ML^{-3}$$
$$V \doteq LT^{-1}$$

The three basic dimensions are required ($r = 3$). Hence, three ($k - r = 6 - 3$) pi terms are to be determined.

We will next select three repeating variables such as $w$, $V$, and $\rho$. Note that it would be incorrect to use both $w$ and $h$ as repeating variables since they have the same dimensions.

Starting with the dependent variable $\mathfrak{D}$, the first pi term can be formed by combining $\mathfrak{D}$ with the repeating variables such that:

$$\Pi_1 = \mathcal{D} w^a V^b \rho^c$$

and in terms of dimensions

$$(MLT^{-2})(L)^a(LT^{-1})^b(ML^{-3})^c = M^0L^0T^0$$

Thus, for $\Pi_1$ to be dimensionless it follows that:

$$\begin{cases} 1+c=0 \\ 1+a+b-3c=0 \\ -2-b=0 \end{cases}$$

and, therefore, $a = -2, b = -2, and\ c = -1.$ The pi term then becomes

$$\Pi_1 = \frac{\mathcal{D}}{w^2V^2\rho}$$

Next the procedure is repeated with the second nonrepeating variable $h$, so that

$$\Pi_2 = hw^aV^b\rho^c$$

It follows that

$$(L)(L)^a(LT^{-1})^b(ML^{-3})^c = M^0L^0T^0$$

$$\Rightarrow \begin{cases} c=0 \\ 1+a+b-3c=0 \\ b=0 \end{cases}$$

so that $a = -1, b = c = 0$, and therefore

$$\Pi_2 = \frac{h}{w}$$

The remaining nonrepeating variable is $\mu$ so that

$$\Pi_3 = \mu w^aV^b\rho^c$$

$$\Rightarrow \quad (ML^{-1}T^{-1})(L)^a(LT^{-1})^b(ML^{-3})^c = M^0L^0T^0$$

$$\Rightarrow \begin{cases} 1+c=0 \\ -1+a+b-3c=0 \\ -1-b=0 \end{cases}$$

and, therefore, $a = -1, b = -1, and\ c = -1.$ The pi term then becomes

$$\Pi_3 = \frac{\mu}{wV\rho}$$

Finally, we can express the results of the dimensional analysis in the form:

$$\frac{\mathcal{D}}{w^2V^2\rho} = f\left(\frac{h}{w}, \frac{\mu}{wV\rho}\right)$$

Or, in the form:

$$\frac{\mathcal{D}}{w^2V^2\rho} = f\left(\frac{w}{h}, \frac{\rho V w}{\mu}\right)$$

which would be more conventional, since the ratio of the plate width $w$ to height $w/h$, is called the aspect ratio, and $\rho V w/\mu$ is the Reynolds number.

The previous dimensional relationship is valid for both prototype and model. Then, for the model:

$$\frac{\mathcal{D}_m}{w_m^2V_m^2\rho_m} = f\left(\frac{w_m}{h_m}, \frac{\rho_m V_m w_m}{\mu_m}\right)$$

The model design conditions, or similarity requirements, are therefore:

$$\frac{w_m}{h_m} = \frac{w}{h} \text{ (geometric similarity)}$$

$$\frac{\rho_m V_m w_m}{\mu_m} = \frac{\rho V w}{\mu} \text{ (kenimatic ans dynamic similarity)}$$

The size of the model is obtained from the first requirement, which indicates that:

$$w_m = \frac{h_m}{h} w$$

We are free to establish the height ratio between the model and prototype, $h_m/h$, but then the model plate width, $w_m$, is fixed in accordance with the previous equation.

The second similarity requirement indicates that the model and prototype must be operated at the same Reynolds number. Thus, the required velocity for the model is obtained from the relationship:

$$V_m = \frac{\mu_m}{\mu}\frac{\rho}{\rho_m}\frac{w}{w_m}V$$

With the foregoing similarity requirements satisfied, the prediction equation for the drag is:

$$\frac{\mathcal{D}}{w^2V^2\rho} = \frac{\mathcal{D}_m}{w_m^2V_m^2\rho_m}$$

or

$$\mathcal{D} = \left(\frac{w}{w_m}\right)^2\left(\frac{\rho}{\rho_m}\right)\left(\frac{V}{V_m}\right)^2\mathcal{D}_m$$

## 5. CHAPTER SUMMARY

This chapter introduces key techniques in fluid mechanics and engineering: dimensional analysis, similitude, and modeling. These methods simplify the analysis of complex systems by reducing them to dimensionless forms, facilitating easier experimental design and predictions.

***Dimensional Analysis***: This method reduces the number of variables in fluid mechanics problems by forming dimensionless groups (pi terms). These groups, such as those based on mass, length, and time, allow engineers to generalize experimental results and focus on key variables. The core principle of dimensional homogeneity states that all terms in a valid physical equation must have consistent dimensions.

***Buckingham Pi Theorem***: The theorem explains how to reduce a dimensionally homogeneous equation with variables to a relationship among independent dimensionless products (pi terms). This is achieved by selecting repeating variables, forming dimensionless groups, and ensuring consistency through experiments.

***Common Dimensionless Groups***: Key groups like Reynolds number (*Re*), Froude number (*Fr*), Euler number (*Eu*), Cauchy number (*Ca*), Strouhal number (*St*), and Weber number (*We*) help characterize fluid systems by comparing different forces (inertial, viscous, gravitational, etc.).

***Modeling and Similitude***: Physical models, designed according to similarity laws, can predict the behavior of prototypes. Similitude includes geometric, kinematic, and dynamic similarity:

Geometric similarity: Ensures proportional dimensions between the model and prototype.

Kinematic similarity: Requires the same velocity and time scale ratios.

Dynamic similarity: Ensures the same force-scale ratio, including Reynolds and Mach numbers, for both model and prototype.

The chapter concludes with example of applying these principles to practical engineering problems, demonstrating how dimensional analysis and similitude can reduce complexity, improve experimental design, and make predictions applicable to real-world systems.

# CONCLUSION

This document has provided a thorough exploration of fluid mechanics, offering both theoretical insights and practical tools for analyzing and understanding fluid motion. Beginning with the study of fluid kinematics, it established the foundation for further exploration by addressing the behavior of fluid particles in motion, using the Eulerian and Lagrangian methods to describe fluid flow. The in-depth analysis of velocity fields, acceleration fields, and various flow patterns provided a strong conceptual basis for understanding the complexities of fluid behavior.

The discussion on potential flow expanded this understanding by introducing the concept of irrotational and incompressible flow, highlighting the role of the velocity potential and stream function in describing fluid motion. The principle of superposition allowed for the creation of more complex flow patterns, making this theory applicable in a variety of engineering contexts.

The section on finite control volume analysis introduced a critical methodology for analyzing fluid systems, where the application of the Reynolds Transport Theorem enabled the examination of mass and momentum exchanges within control volumes. This is crucial for practical applications in systems such as pumps, turbines, and jet engines, where understanding the forces and behavior of fluids is essential for design and operation.

Additionally, the incorporation of dimensional analysis and similitude provided invaluable tools for simplifying complex systems, making it easier to derive generalized solutions and create scalable models. This is especially beneficial in fluid dynamics, where experimental setups are often employed to complement theoretical models and improve predictive accuracy.

In summary, this fundamental course provides a solid foundation in fluid mechanics, preparing students for more advanced studies in the field. By mastering the core principles and analytical techniques presented, engineers and researchers are equipped to tackle complex fluid flow problems in various industries. This course serves as a stepping stone toward deeper exploration and application of fluid mechanics in advanced engineering challenges.

# RECOMMENDED TEXTBOOKS

**[1]** Bruce R. Munson, Theodore H. Okiishi, Wade W. Huebsch, and Alric P. Rothmayer. Fundamentals of Fluid Mechanics, 7th Edition, John Wiley & Sons, Inc. (2013)

**[2]** Frank M White. Fluid mechanics, 5th Edition -McGraw-Hill (2003)

**[3]** Yunus A. Çengel and John M. Cimbala. Fluid Mechanics fundamentals and applications, 4th Edition, McGraw-Hill (2020)

**[4]** Merle C. Potter, David C. Wiggert, Bassem Ramadan, and Tom I-P. Shih. Mechanics of Fluids, 4th Edition, Cengage Learning (2012)

**[5]** Pijush K. Kundu, Ira M. Cohen, David R. Dowling, Jesse Capecelatro. Fluid mechanics, 7th Edition, Academic Press (2025)