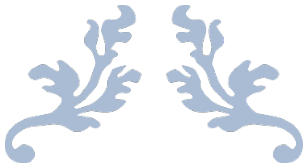


NOTES ON TRANSPORT PHENOMENA

R Shankar Subramanian

UPDATED AUGUST 1, 2020

DEPARTMENT OF CHEMICAL AND BIOMOLECULAR ENGINEERING, CLARKSON UNIVERSITY
Potsdam, New York 13699-5705

Introductory Remarks

These notes were developed to support an introductory graduate level course on transport phenomena at Clarkson University that I taught for many years, and later also used to supplement material from the textbook by R.B. Bird, W.E. Stewart, and E.N. Lightfoot titled "Transport Phenomena" in my undergraduate elective course on the subject. The notes should not be construed as complete in the sense of being useful as a textbook for any course. Rather, they contain a variety of sections that I wrote over the years to provide supplemental material for students taking my courses, and only reflect my style and bias in presenting topics.

Because of the manner in which these notes were formulated and modified over the last three decades, the various parts were written nearly independently of each other, and each section contains its own set of references.

In preparing and refining these notes, I have learned much from the students who took my courses as well as colleagues at Clarkson and elsewhere, to whom I shall remain grateful forever.

Any errors in this document are entirely my responsibility. I can be reached at

R. Shankar Subramanian
Professor Emeritus
Department of Chemical and Biomolecular Engineering
Clarkson University
Potsdam, New York 13699
subramanian@clarkson.edu

July 2020
Update 1, August 2020 (minor edits in Section II)

Table of Contents

Part I

Introduction to Vectors and Tensors

Introduction to Vectors and Tensors

Some useful references for learning about vectors and tensors are the books listed as references at the end.

Some Basics

We encounter physical entities such as position, velocity, momentum, stress, temperature, heat flux, concentration, and mass flux in transport problems - there is a need to describe them in mathematical terms and manipulate the representations in various ways. This requires the tools of tensor analysis.

Scalars

An entity such as temperature or concentration that has a magnitude (and some units that need not concern us right now), but no sense of direction, is represented by a scalar.

Vectors

In contrast, consider the velocity of a particle or element of fluid; to describe it fully, we need to specify both its magnitude (in some suitable units) and its instantaneous spatial direction. Other examples are momentum, heat flux, and mass flux. These quantities are described by vectors. In books, vectors are printed in boldface. In ordinary writing, we may represent a vector in different ways.

$\underline{v}, \vec{v}, \vec{v}$ or v_i

Gibbs notation **index notation**

The last requires comment. Whereas we represent the vectorial quantity with a symbol, we often know it only via its components in some basis set. Note that the vector as an entity has an **invariant** identity independent of the basis set in which we choose to represent it.

In index notation, the subscript "i" is a free index - that is, it is allowed to take on any of the three values 1, 2, 3, in 3-dimensional space. Thus, v_i really stands for the ordered set (v_1, v_2, v_3).

Basis Sets

The most common basis set in three-dimensional space is the orthogonal triad $(\underline{i}, \underline{j}, \underline{k})$ corresponding to a rectangular Cartesian coordinate system. $\underline{i}$ stands for a unit vector in the $x-$ direction and $\underline{j}$ and $\underline{k}$ represent unit vectors in the y and $z-$directions respectively. Note that this is not a unique basis set. The directions of $\underline{i}, \underline{j}, \underline{k}$ depend on our choice of the coordinate directions.

There is no reason for the basis set to be composed of orthogonal vectors. The only requirement is that the three vectors chosen do not lie in a plane. Orthogonal sets are the most convenient, however.

We find the components of a vector in the directions of the base vectors by taking inner (dot) products.

$$v_x = \underline{v} \cdot \underline{i}, \quad v_y = \underline{v} \cdot \underline{j}, \quad v_z = \underline{v} \cdot \underline{k}$$

Then, $\underline{v} = v_x \underline{i} + v_y \underline{j} + v_z \underline{k}$

You can verify the consistency of the above by taking inner products of both sides of the equation with the base vectors and recognizing that the base vectors are orthogonal.

$$\underline{i} \cdot \underline{j} = \underline{j} \cdot \underline{k} = \underline{k} \cdot \underline{i} \equiv 0$$

The order of the vectors in the inner product is unimportant.

$$\underline{a} \cdot \underline{b} = \underline{b} \cdot \underline{a}$$

Scalar and Vector Fields

In practice the temperature, velocity, and concentration in a fluid vary from point to point (and often with time). Thus, we think of fields - temperature field, velocity field, etc.

In the case of a vector field such as the velocity in a fluid, we need to represent the velocity at every point in space in the domain of interest. The advantage of the rectangular Cartesian basis set $\left(\underline{i}, \underline{j}, \underline{k}\right)$ is that it is invariant as we translate the triad to any point in space. That is, not only are these base vectors of unit length, but they never change direction as we move from one point to another, once we have chosen our $x, y,$ and z directions.

Vector Operations

The entity $\underline{v}$ has an identity of its own. Its length and spatial direction are independent of the basis set we choose. As the vectors in the basis set change, the **components** of $\underline{v}$ change according to standard rules.

Vectors can be added; the results are new vectors. If we use component representation, we simply add each component. Subtraction works in a similar manner.

Vectors also can be multiplied, but there is more than one way to do it. We define the dot and cross products, also known as **inner (**or **scalar)** and **vector** products, respectively, as shown below.

$\underline{a} \cdot \underline{b} = a_x b_x + a_y b_y + a_z b_z$ is a scalar. We commonly use a numerical subscript for the components;

in this case, the basis set is the orthogonal triad $\left(\underline{e}_{(1)}, \underline{e}_{(2)}, \underline{e}_{(3)}\right)$. Let

$$\underline{a} = a_1\underline{e}_{(1)} + a_2\underline{e}_{(2)} + a_3\underline{e}_{(3)}$$

Then,

$$\underline{a}\cdot\underline{b} = a_1b_1 + a_2b_2 + a_3b_3$$

$$= \sum_{i=1}^{3} a_ib_i$$

In the above, we usually omit $\sum_{i=1}^{3}$. When an index is repeated, summation over that index is implied.

$\underline{a}\cdot\underline{b} = a_ib_i$ This is called the Einstein summation convention

$\underline{a}\cdot\underline{a} = a_ia_i = a^2$ or $|\underline{a}|^2$

where a is the length of $\underline{a}$ and is **invariant**; "invariant" means that the entity does not change as the basis set is altered.

$\underline{a}\times\underline{b}$ is the vector product. As implied by the name, it is a vector; it is normal to the plane containing $\underline{a}$ and $\underline{b}$. $(\underline{a}, \underline{b}, \underline{a}\times\underline{b})$ form a right-handed system (this is an arbitrary convention , but we have to choose one or the other, so we choose "right"). The order is important, for,

$$\underline{a}\times\underline{b} = -\underline{b}\times\underline{a}$$

that is, $\underline{b}\times\underline{a}$ points opposite to $\underline{a}\times\underline{b}$.

We can write

$$\underline{a}\times\underline{b} = \begin{vmatrix} \underline{e}_{(1)} & \underline{e}_{(2)} & \underline{e}_{(3)} \\ a_1 & a_2 & a_3 \\ b_1 & b_2 & b_3 \end{vmatrix}$$

There is a compact representation of a determinant that helps us write

$$\underline{a}\times\underline{b} = \varepsilon_{ijk}\, a_ib_j$$

(Note that k is a free index. The actual symbol chosen for it is not important; what matters is that the right side has **one** free index, making it a vector)

ε_{ijk} is called the permutation symbol

$\varepsilon_{ijk} = 0$ if any two of the indices are the same

$= +1$ if i, j, k form an even permutation of 1, 2, 3 [example: 1,2,3]

$= -1$ if i, j, k form an odd permutation of 1, 2, 3 [example: 2, 1, 3]

We can assign a geometric interpretation to $\underline{a} \cdot \underline{b}$ and $\underline{a} \times \underline{b}$. If the angle between the two vectors $\underline{a}$ and $\underline{b}$ is θ, then

$$\underline{a} \cdot \underline{b} = a\ b \cos\theta$$

and the length of $\underline{a} \times \underline{b}$ is $a\,b \sin\theta$. You may recognize $a\ b \sin\theta$ as the area of the parallelogram formed by $\underline{a}$ and $\underline{b}$ as two adjacent sides. Given this, it is straightforward to see that

$$\underline{a} \cdot \underline{b} \times \underline{c} = \varepsilon_{ijk}\, a_i b_j c_k$$

is the volume of the parallelepiped with sides $\underline{a}, \underline{b}, \text{and } \underline{c}$. This is called the triple scalar product.

Second Order Tensors

Note that we did not define vector division. The closest we come is in the definition of second-order tensors!

Imagine

$$\frac{\underline{a}}{\underline{b}} = \underline{\underline{T}}$$

Instead, we write

$$\underline{a} = \underline{\underline{T}} \cdot \underline{b}$$

A tensor (unless explicitly stated otherwise we'll only be talking about "second-order" and shall therefore omit saying it every time) "operates" on a vector to yield another vector. It is very useful to think of tensors as operators as you'll see later.

Note the "dot" product above. Using ideas from vectors, we can see how the above equation may be written in index notation.

$$a_i = \mathrm{T}_{ij} b_j$$

It is important to note that $\underline{b} \cdot \underline{\underline{\mathrm{T}}}$ would be $b_i \mathrm{T}_{ij}$ and would be different from $\underline{\underline{\mathrm{T}}} \cdot \underline{b}$ in general.

The two underbars in $\underline{\underline{\mathrm{T}}}$ now take on a clear significance; we are referring to a doubly subscripted entity. We can think of a tensor as a sum of components in the same way as a vector. For this, we use the following result.

$\underline{e}_{(i)} \cdot \underline{\underline{\mathrm{T}}} \cdot \underline{e}_{(j)} = \mathrm{T}_{ij}$ Scalar

We're not using index notation on the left side of the above equality.

Thus, to get T_{23} we would find $\underline{e}_{(2)} \cdot \underline{\underline{\mathrm{T}}} \cdot \underline{e}_{(3)}$. We can then think of T as a sum.

$$\underline{\underline{\mathrm{T}}} = \mathrm{T}_{11}\, \underline{e}_{(1)} \underline{e}_{(1)} + \mathrm{T}_{12}\, \underline{e}_{(1)} \underline{e}_{(2)} + \mathrm{T}_{13}\, \underline{e}_{(1)} \underline{e}_{(3)}$$

$$+ \mathrm{T}_{21}\, \underline{e}_{(2)} \underline{e}_{(1)} + \mathrm{T}_{22}\, \underline{e}_{(2)} \underline{e}_{(2)} + \mathrm{T}_{23}\, \underline{e}_{(2)} \underline{e}_{(3)}$$

$$+ \mathrm{T}_{31}\, \underline{e}_{(3)} \underline{e}_{(1)} + \mathrm{T}_{32}\, \underline{e}_{(3)} \underline{e}_{(2)} + \mathrm{T}_{33}\, \underline{e}_{(3)} \underline{e}_{(3)}$$

What are the quantities $\underline{e}_{(1)} \underline{e}_{(2)}$ and others like them? They are called **dyads**. They are a basis set for representing tensors. Each is a tensor that only has one component in this basis set. Note that $\underline{e}_{(i)} \underline{e}_{(j)} \neq \underline{e}_{(j)} \underline{e}_{(i)}$. A dyadic product is a third way in which we can multiply two vectors.

You can see that tensors and matrices have a lot in common!

In fact, we commonly write the components of a tensor as the elements of a 3 x 3 matrix.

$$\begin{pmatrix} \mathrm{T}_{11} & \mathrm{T}_{12} & \mathrm{T}_{13} \\ \mathrm{T}_{21} & \mathrm{T}_{22} & \mathrm{T}_{23} \\ \mathrm{T}_{31} & \mathrm{T}_{32} & \mathrm{T}_{33} \end{pmatrix}$$

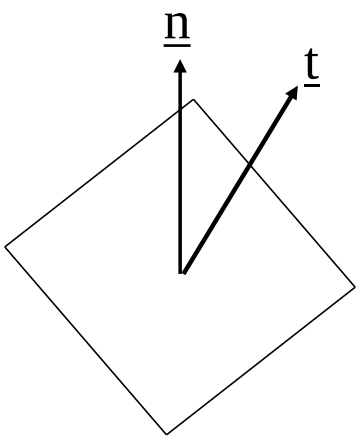
Naturally, as we change our basis set, the components of a given tensor will change, but the entity itself does not change. Of course, unlike vectors, we cannot visualize tensors – we only "know" them by what they do to vectors that we "feed" them!

A good example of a tensor in fluid mechanics is the stress at a point. To completely specify the stress vector, we not only need to specify the point, but also the orientation of the area element. At a given point, we can orient the area in infinitely many directions, and for each orientation, the stress vector would, in general be different.

$$\text{Stress} = \frac{\text{Force}}{\text{Area}} \quad \begin{matrix} \leftarrow \text{has magnitude and direction} \\ \leftarrow \text{has magnitude and direction} \end{matrix}$$

In fact, we can show that stress is indeed a tensor (for proof, see Aris, p. 101). So, we get

$$\underline{t} = \underline{n} \cdot \underline{\underline{\mathrm{T}}}$$



The symbol $\underline{n}$ represents the unit normal (vector) to the area element, and $\underline{t}$ is the stress vector acting on that element. The second-order tensor $\underline{\underline{\mathrm{T}}}$ completely describes the state of stress at a point. By convention, $\underline{t}$ is the stress exerted **by** the fluid into which $\underline{n}$ points **on** the fluid adjoining it.

Just as a vector has one invariant (its length), a tensor has three invariants. They are defined as follows.
Let $\underline{\underline{A}}$ or A_{ij} be the tensor.

$$\text{abbreviation} \downarrow$$

$$I_A = \text{trace}\left\{\underline{\underline{A}}\right\} = tr\left\{\underline{\underline{A}}\right\} = A_{ii}$$

$$II_A = \frac{1}{2}\left[\left|I_A\right|^2 - \overline{II}_A\right] \text{ where}$$

$$\overline{II}_A = tr\left\{\underline{\underline{A}} \cdot \underline{\underline{A}}\right\} \qquad \text{Note: } \underline{\underline{A}} \cdot \underline{\underline{A}} \text{ is the tensor } A_{ij}A_{jk}$$

$$III_A = \text{ Determinant of } \underline{\underline{A}} = \text{Det}\left\{\underline{\underline{A}}\right\}$$

$$= \varepsilon_{ijk} A_{1i} A_{2j} A_{3k}$$

$$= \varepsilon_{ijk} A_{il} A_{j2} A_{k3}$$

As the basis set is changed, the invariants do not change even though the components of the tensor may change. For more details, consult Aris, p. 26, 27 or Slattery, p. 47, 48.

A **symmetric** tensor A_{ij} is one for which $A_{ij} = A_{ji}$. Thus, there are only six independent components. Stress is a symmetric tensor (except in unusual fluids). Symmetric tensors with real elements are self-adjoint operators, a concept about which you can learn more in advanced work.

A **skew-symmetric** tensor A_{ij} is one for which $A_{ij} = -A_{ji}$. You can see immediately that the diagonal elements must be zero $\left(\text{because } A_{ii} = -A_{ii}\right)$. Skew-symmetric tensors have only three independent components. **Vorticity** is an example of a skew symmetric tensor.

If we write a skew-symmetric tensor A_{ij} in the form

$$\begin{pmatrix} 0 & a_3 & -a_2 \\ -a_3 & 0 & a_1 \\ a_2 & -a_1 & 0 \end{pmatrix}$$

We can see that there is a vector $\underline{a}$ that can be formed using the elements of A_{ij}. The two are related by the following result, which is useful in the context of the physical significance of vorticity.

$$\underline{\underline{A}} \cdot \underline{x} = \underline{x} \times \underline{a}$$

Any second order tensor can be decomposed into the sum of a symmetric tensor and a skew-symmetric tensor.

$$A_{ij} = \underbrace{\frac{1}{2}\left(A_{ij} + A_{ji}\right)}_{\text{Symmetric Tensor}} + \underbrace{\frac{1}{2}\left(A_{ij} - A_{ji}\right)}_{\text{Skew-Symmetric Tensor}} \quad \text{or in Gibbs notation, } \underline{\underline{A}} = \underbrace{\frac{1}{2}\left(\underline{\underline{A}} + \underline{\underline{A}}^T\right)}_{\text{Symmetric Tensor}} + \underbrace{\frac{1}{2}\left(\underline{\underline{A}} - \underline{\underline{A}}^T\right)}_{\text{Skew-Symmetric Tensor}}$$

Here, $\underline{\underline{A}}^T$ is the transpose of the tensor $\underline{\underline{A}}$. $\underline{\underline{A}}^T$ has components that form a matrix whose columns are the rows of the matrix of components of $\underline{\underline{A}}$.

There is a special tensor that leaves a vector undisturbed. It is called the identity or unit tensor $\underline{\underline{I}}$.

$$\underline{\underline{I}} = \begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{pmatrix} \qquad \underline{\underline{I}} \cdot \underline{x} = \underline{x} \quad \text{for any } \underline{x}$$

In index notation, we write $\underline{\underline{I}}$ as δ_{ij}, the Kronecker delta.

Symmetric tensors have a very special property. Remember that we define a tensor as the representation of some field at a point; at that point, there are **three special directions, orthogonal to each other**, associated with a symmetric tensor. When the tensor operates on a vector in one of these directions, it returns another vector pointing in the same (or exactly opposite) direction! The new vector, however, will have a different length in general. This multiplication factor in the length is called the **principal value** or **eigenvalue** of the tensor. If the eigenvalue is negative, the output vector from the operation will point opposite to the input vector.

Because there are, in general, three directions that are special, there are usually three distinct principal values, one associated with each direction. Even for tensors that are not symmetric, there are three principal values; however these need not all be real. Sometimes, two are complex. Even when the principal values are real, the directions associated with them need not be orthogonal if the tensor is not symmetric.

The problem for the principal or eigenvalues of $\underline{\underline{A}}$ is

$$\underline{\underline{A}} \cdot \underline{x} = \lambda \underline{x} \equiv \lambda \underline{\underline{I}} \cdot \underline{x}$$

Therefore,

$$\left[\underline{\underline{A}} - \lambda \underline{\underline{I}} \right] \cdot \underline{x} = \underline{0}$$

From linear algebra, for non-trivial solutions of the above system to exist, we must have

$$\det\left[\underline{\underline{A}} - \lambda \underline{\underline{I}} \right] = 0$$

The resulting third degree equation for the eigenvalues is

$$-\lambda^3 + I_A \lambda^2 - II_A \lambda + III_A = 0$$

and has three roots λ_1, λ_2, and λ_3. When these roots are each used, in turn, and we solve for $\underline{x}$, we obtain an **eigenvector** that is known only to within an arbitrary multiplicative constant. Commonly, the eigenvector is normalized so that it has unit length.

From the above, you can see that corresponding to a symmetric tensor, there is a special rectangular Cartesian set of basis vectors of unit length. If we choose this as the basis set, the tensor will have a simple diagonal form with the diagonal components being the eigenvalues.

If you're wondering what happens when two eigenvalues are identical, it is easy to show that any vector in the plane normal to the third eigenvector (corresponding to the third eigenvalue) is acceptable as an eigenvector. In other words, on that plane, the tensor operating on a vector in any direction will yield a vector in the same direction with a magnification factor corresponding to the repeated eigenvalue.

If all three eigenvalues are identical, then any direction in space will be acceptable as the direction of the eigenvectors. Such a tensor is called **isotropic** for this reason. $\underline{\underline{I}}$ is an isotropic tensor with eigenvalues equal to unity. Any scalar multiple of $\underline{\underline{I}}$ also is isotropic.

Vector Calculus

If we consider a scalar field such as temperature, we find the rate of change with distance in some direction, x, by calculating $\partial \mathrm{T}/\partial x$. How can we represent the rate of change in three-dimensional space without specifying a particular direction? We do this via the **gradient operator**. The entity $\nabla \mathrm{T}$ [we call it "grad T"] is a vector field. In index notation we write the gradient operator as $\partial / \partial x_i$ where i is a free index, so that $\nabla \mathrm{T} = \partial T / \partial x_i$.

At a given point in space, the vector $\nabla \mathrm{T}$ points in the direction of greatest change of T. To obtain the rate of change of T at that point in any specified direction, $\underline{n}$, we simply "project" $\nabla \mathrm{T}$ in that direction.

$$\frac{\partial \mathrm{T}}{\partial n} = \nabla \mathrm{T} \cdot \underline{n}$$

unit vector

Surfaces in space on which a field has the same value everywhere are level surfaces. In the case of temperature fields, these surfaces are called **isotherms**. Along such a surface, the temperature cannot change. Therefore, the $\nabla \mathrm{T}$ vector is everywhere **normal** to isothermal surfaces since it must yield a value of zero when projected onto such surfaces.

It is straightforward to establish from the definition that

$$\nabla \equiv \underline{e}_{(1)} \frac{\partial}{\partial x_1} + \underline{e}_{(2)} \frac{\partial}{\partial x_2} + \underline{e}_{(3)} \frac{\partial}{\partial x_3}$$

in a rectangular Cartesian coordinate system (x_1, x_2, x_3).

Note that ∇ is an operator and not a vector. So, you should exercise care in manipulating it.

The ∇ operator is the generalization of a derivative. We can differentiate vector fields in more than one way.

Divergence

$\nabla \cdot \underline{v}$ or div $\underline{v}$ is called the divergence of the vector field $\underline{v}$. If the rectangular Cartesian components of $\underline{v}$ are v_1, v_2, v_3, then

$$\nabla \cdot \underline{v} = \frac{\partial v_1}{\partial x_1} + \frac{\partial v_2}{\partial x_2} + \frac{\partial v_3}{\partial x_3} = \frac{\partial v_i}{\partial x_i} \text{ in index notation}$$

As you can see, the result is a scalar field. It can be shown that the divergence of a vector field at any point represents the outflow of the field.

Curl

$\nabla \times \underline{v}$ or curl $\underline{v}$ is a vector field. As the name implies, it measures the "rotation" of the vector $\underline{v}$. Again, in $\left(x_{1,} x_2, x_3\right)$ coordinates,

$$\nabla \times \underline{v} = \begin{vmatrix} \underline{e}_{(1)} & \underline{e}_{(2)} & \underline{e}_{(3)} \\ \dfrac{\partial}{\partial x_1} & \dfrac{\partial}{\partial x_2} & \dfrac{\partial}{\partial x_3} \\ v_1 & v_2 & v_3 \end{vmatrix} = \varepsilon_{ijk} \frac{\partial v_k}{\partial x_j} \text{ in index notation.}$$

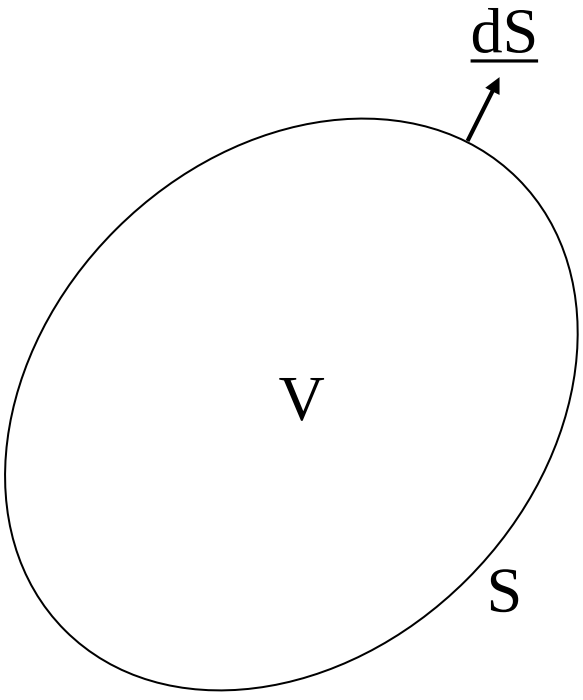
The term "curl" implies rotation, and the curl of a vector field is indeed related to rotation as will be evident when we discuss fluid mechanics later.

There are two important theorems you should know. They are simply stated here without proof.

Divergence Theorem

If the volume V in space is bounded by the surface S,

$$\int_V \nabla \cdot \underline{a}\, dV = \int_S \underline{dS} \cdot \underline{a}$$



The vector field $\underline{a}$ should be continuous and differentiable. The symbol $\underline{dS}$ represents a vector surface element. If $\underline{n}$ is the unit normal to the surface,

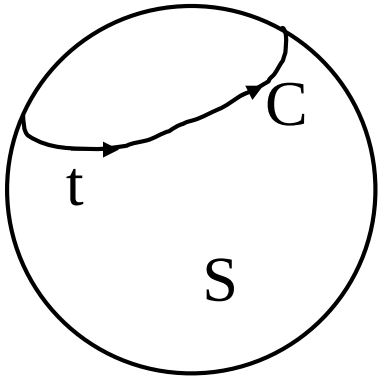
$$\underline{dS} = \underline{n}\, dS$$

The entity dV is a volume element. In the theorem, the left side is a volume integral and the right side is an integral over the surface that bounds the volume. Finally, $\underline{a}$ need not be a vector field, but can be a tensor field of any order.

The **divergence theorem**, also known as **Green's transformation,** is a very useful result that permits us to convert volume integrals into surface integrals. By applying it to an infinitesimal volume, you can visualize the physical significance of the divergence of a vector field at a point as the outward "flow" of the field from that point.

Stokes Theorem

This permits the conversion of integrals over a surface to those around a bounding curve. Imagine a surface that does not completely enclose a volume, but rather is open, such as a baseball cap. Let S be the surface and C, the curve that bounds it.

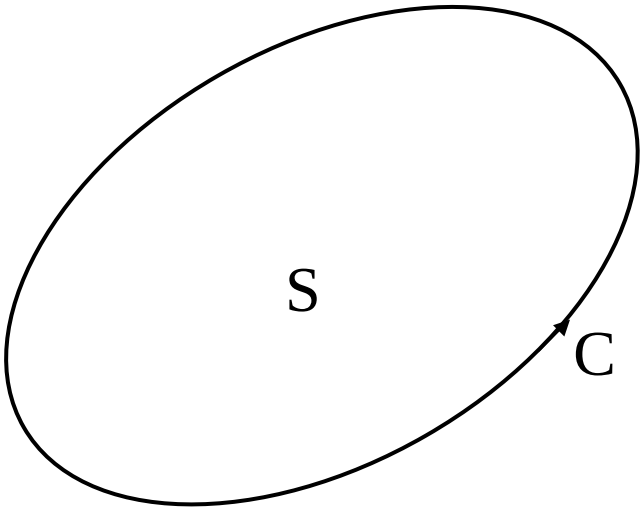


If a vector field $\underline{a}$ is defined everywhere necessary, and is continuous and differentiable, Stokes theorem states:

$$\int_S \left(\nabla \times \underline{a}\right) \cdot \underline{dS} = \int_C \underline{a} \cdot \underline{t}\, ds$$

where $\underline{dS}$ is a vector area element on the surface S, ds is a scalar line element on the bounding curve C, and t is a unit tangent vector on C.

The integral on the right side is known as the **circulation** of $\underline{a}$ around the closed curve C. The field $\underline{a}$ appearing in the theorem can be replaced by a tensor field of any desired order.



By imagining the surface S to lie completely on the plane of the paper as shown, you can visualize the physical significance of $\nabla \times \underline{a}$. If you make S shrink to an infinitesimal area, the area integral on the left side becomes the product of the component of $\nabla \times \underline{a}$ normal to the plane of the paper and the area. The line integral is still the circulation around an infinitesimal closed loop surrounding the point. If $\underline{a}$ is the velocity field $\underline{v}$, by making the boundary an infinitesimal circle of radius ε, the right side can be seen to be approximately $2\pi\varepsilon v$, where v is the magnitude of the velocity around the loop. The left side is approximately $\pi\varepsilon^2 (\nabla \times v) \cdot \underline{n}$ where $\underline{n}$ is the unit normal to the plane of the paper. Therefore, $\frac{1}{2}(\nabla \times \underline{v}) \cdot \underline{n} \approx \frac{v}{\varepsilon}$, which becomes the instantaneous angular velocity of the fluid at the point on the plane of the paper as $\varepsilon \to 0$. Because there is nothing unique about the choice of the plane of the paper, we can see that $\frac{1}{2}(\nabla \times \underline{v})$ in fact represents the instantaneous angular velocity vector of a fluid element at a given point, the component of which in any direction is obtained by projecting in that direction.

The Gradient of a Vector Field

Just as we defined the gradient of a scalar field, it is possible to define the gradient of a vector or tensor field. If $\underline{v}$ is a vector field, $\nabla\underline{v}$ is a second-order tensor field. The rate of change of $\underline{v}$ in any direction $\underline{n}$ is given by

$$\frac{\partial \underline{v}}{\partial n} = \nabla\underline{v} \cdot \underline{n}$$

Concluding Remark

In this section of the notes, we have used symbols for vectors and tensors that are convenient for handwritten work. In the remainder of these notes, we shall use the standard boldface representation of vectors and tensors found in books.

References

1. R. Aris, Vectors, Tensors, and the Basic Equations of Fluid Mechanics, Dover Publications, 1989 (original by Prentice-Hall, 1962).

2. R. B. Bird, W. E. Stewart, and E. N. Lightfoot, Transport Phenomena, John Wiley, 2007.

3. J. Slattery, Momentum, Energy, and Mass Transfer in Continua, McGraw-Hill, 1972.

Part II

Continuity Equation: Principle of Conservation of Mass

Kinematics of Fluid Motion

Navier-Stokes Equation: Principle of Conservation of Momentum

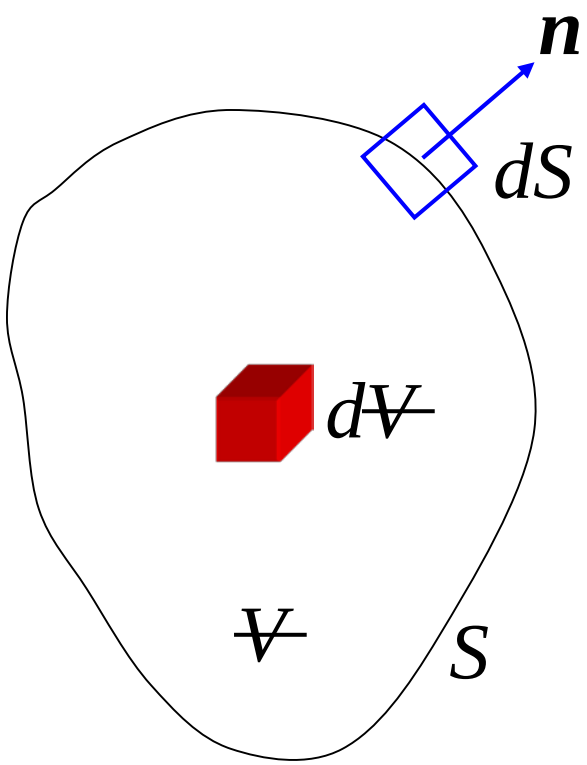
Procedure for Setting up Problems

Boundary Conditions

Continuity Equation: Principle of Conservation of Mass

Based on observation, one can postulate the idea that mass is neither created nor destroyed. In other words, it is conserved. This is termed the Principle of Conservation of Mass. This principle is applied to a fixed volume of arbitrary shape in space that contains fluid. This volume is called a "Control Volume." Fluid is permitted to enter or leave the control volume.

A control volume $\forall$ is shown in the sketch.



Also marked on the sketch is the bounding surface S of this control volume, called the control surface; an element of surface area dS and the unit outward normal (vector) to that area element, $\boldsymbol{n}$ are shown as well. The vector symbol $\boldsymbol{dS} = \boldsymbol{n}\, dS$ is used to represent a directed differential area element on the surface.

Just like the principle of conservation of mass, one can make similar statements about energy and momentum, being careful to accommodate ways in which energy or momentum can enter or leave a fixed volume in space occupied by a fluid. These conservation statements are put in mathematical form and termed "integral balances." These balances are useful in a variety of problems. Here, we shall apply the principle of conservation of mass to the control volume shown in the sketch, and eventually obtain a partial differential equation commonly known as the continuity equation. We begin with a verbal statement of the principle of conservation of mass.

Rate of increase of mass of material within the control volume = Net rate at which mass enters the control volume.

Let us write a mathematical representation of the above statement. To determine the rate of accumulation of mass in the control volume, we begin with the mass content in the differential volume element $d\forall$, because the density ρ of the fluid can depend on position. Multiplying the differential volume by the density at that location gives the amount of mass in the differential volume element, and the total mass M in the control volume $\forall$ is obtained by integrating this product over the entire control volume. Therefore,

$$M = \int_{\forall} \rho \, d\forall$$

The time rate of change of this mass content in the control volume is dM / dt. Because the control volume is fixed in space, the time derivate can be taken inside the integral and becomes a partial derivate in time, obtained when keeping spatial coordinates fixed. Thus, the left side in the verbal statement of the principle of conservation of mass is $\int_{\forall} \frac{\partial \rho}{\partial t} d\forall$, where t represents time.

Now, we need to develop a result for the net rate of entry of mass into the control volume through the control surface. For this, we consider the differential area element dS. Now, let us define the mass flux through space as the vector $\boldsymbol{N}_M$. Then the rate of entry of mass into the control volume through the area element dS is $\left(-\boldsymbol{N}_M \bullet \boldsymbol{n}\, dS\right)$, which is $\left(-\boldsymbol{N}_M \bullet \boldsymbol{dS}\right)$. The total rate of entry of mass over the entire surface can be written as the integral $\int_S \left(-\boldsymbol{N}_M \bullet \boldsymbol{dS}\right)$. This is the right side in the verbal statement of the principle of conservation of mass. Equating the two sides yields the following result.

$$\int_{\forall} \frac{\partial \rho}{\partial t} d\forall = -\int_S \boldsymbol{N}_M \bullet \boldsymbol{dS} \quad \text{or} \quad \boxed{\int_{\forall} \frac{\partial \rho}{\partial t} d\forall + \int_S \boldsymbol{N}_M \bullet \boldsymbol{dS} = 0}$$

A theorem that applies to vector fields permits us to convert a surface integral such as the one in the above equation into a volume integral. It is known as the Gauss divergence theorem or Green's theorem. The vector field must satisfy conditions regarding continuity of derivatives, and all the fields that we encounter are assumed to satisfy these conditions. Using this theorem, we can write the following result.

$$\int_S \boldsymbol{N}_M \bullet \boldsymbol{dS} = \int_{\forall} \left(\nabla \bullet \boldsymbol{N}_M\right) d\forall$$

Using this equality, we can rewrite the principle of conservation of mass as

$$\int_{\forall} \left[\frac{\partial \rho}{\partial t} + \nabla \bullet \boldsymbol{N}_M \right] d\forall = 0$$

We wish to conclude that at every point in the fluid, the integrand of the above result must be zero. To do this we need to use two important ideas. One is that our control volume is arbitrary in shape and location. The second is that the fields ρ, $\boldsymbol{N}_M$, and their derivatives are all continuous functions of position. Thus, the integrand is a continuous function of position. This means that if the integrand is non-zero at any point in space, we are guaranteed a neighborhood of that point in which it retains the same sign. Then, we can consider that specific neighborhood the control volume, in which case the integral will be non-zero, violating the integral balance stated above. This precludes the integrand being non-zero anywhere in the fluid. Thus, it must be zero at every point in the fluid.

$$\frac{\partial \rho}{\partial t} + \nabla \bullet \boldsymbol{N}_M = 0$$

The mass flux is the product of the density and the volume flux, which is the velocity.

$$\boldsymbol{N}_M = \rho \boldsymbol{v}$$

Substituting this result in the partial differential equation yields the continuity equation.

$$\frac{\partial \rho}{\partial t} + \nabla \bullet \left(\rho \boldsymbol{v} \right) = 0$$

By working out the divergence of the product, we can rewrite this as

$$\frac{\partial \rho}{\partial t} + \boldsymbol{v} \bullet \nabla \rho + \rho \left(\nabla \bullet \boldsymbol{v} \right) = 0$$

It is common practice to combine the first two terms in the left side and write the result as the material derivative of density with respect to time.

$$\frac{d\rho}{dt} = \frac{\partial \rho}{\partial t} + \boldsymbol{v} \bullet \nabla \rho$$

Bird et al. (1) use the symbol $\frac{D\rho}{Dt}$ for the material derivative. We shall use $\frac{d\rho}{dt}$ in our work. The material derivative is the time derivative taken while keeping the material coordinates fixed. Physically, this means that it is the time derivative obtained while moving with a material particle, i.e. moving with the flow.

Thus, the continuity equation is rewritten as

$$\frac{d\rho}{dt} + \rho\left(\nabla \bullet \boldsymbol{v}\right) = 0$$

The assumption of incompressible flow, implying that the density of an element of fluid does not change with a change in pressure, is used commonly. With this assumption, the continuity equation reduces to

$$\boxed{\nabla \bullet \boldsymbol{v} = 0}$$

In rectangular Cartesian coordinates (x, y, z) this is written as follows.

$$\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0$$

Appropriate representations in other coordinate systems can be found in textbooks such as that by Bird et al. (1).

The Assumption of Incompressible Flow

Incompressible flow implies that the variation in the density of an element of fluid due to changes in pressure can be considered negligible. Because pressure variations are encountered in fluids, one might wonder about the limits on the validity of this assumption. Liquids are virtually incompressible, displaying very small variations in density in response to pressure changes, so that incompressible flow is almost always an excellent assumption in liquids. Gases, on the other hand, undergo a significant density change when the pressure is changed. In the flow of gases, the assumption of incompressible flow can be used so long as

$$\left|\frac{\Delta\rho}{\rho}\right| \ll 1$$

where $\Delta\rho$ represents a typical change in density encountered in the flow, and the symbol $\ll$ stands for "much less than."

As shown in pages 9-11 of Schlichting (2) this condition implies that the square of the ratio of a characteristic velocity in the fluid v_0 to the speed of sound c in the fluid, is much less than unity.

$$\left(\frac{v_0}{c}\right)^2 \ll 1$$

The Mach number M is defined as

$$M = \frac{v_0}{c}$$

so that the requirement is that $M^2 \ll 1$. In practical terms, this means that the assumption of incompressible flow in a gas is good so long as the Mach number is relatively small, that is, so long as the typical flow velocities are much smaller than the speed of sound in the fluid.

A more detailed discussion of the conditions for the validity of the assumption of incompressible flow, accounting also for thermal expansion, can be found in pages 167-171 of Batchelor (3).

References

1. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, Transport Phenomena, Wiley, 2007.

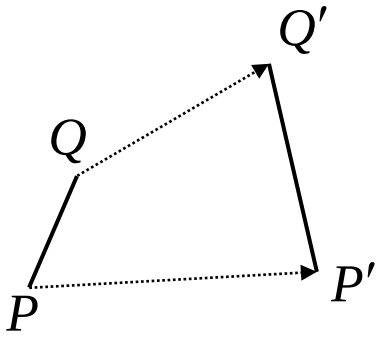
2. H. Schlichting, Boundary Layer Theory, McGraw-Hill, 1968.

3. G. K. Batchelor, An Introduction to Fluid Dynamics, Cambridge University Press, 1967.

Kinematics of Fluid Motion

Kinematics is the study of motion without dealing with the forces that affect motion. The discussion here is of limited scope and for more details, the reader is encouraged to consult any of the references listed at the end. The notation used and the details of the development in many places are directly borrowed from Aris (1) and Batchelor (2). Our focus here is on fluid motion. We shall use rectangular Cartesian coordinates (x, y, z), along with the associated basis set of mutually orthogonal unit vectors $(\boldsymbol{i}, \boldsymbol{j}, \boldsymbol{k})$. The position vector is labeled $\boldsymbol{x}$.

Imagine a tiny line element $d\boldsymbol{x}$, labeled PQ in the sketch, at some instant of time. After a small amount of time dt, the two ends have moved to new locations because of fluid motion, and the new line element is labeled $P'Q'$.



We can see that if the velocity were to be the same at both ends of the element, it would change neither its length, nor its orientation. Therefore, in a uniform velocity field, there is simple translation of fluid elements with no deformation or rotation. To cause either, the velocity $\boldsymbol{v}(\boldsymbol{x})$ must be non-uniform. To understand the nature of the changes in fluid elements brought about by the flow, we must, therefore, investigate the velocity gradient, $\nabla\boldsymbol{v}$, which is a second order tensor.

From calculus, we know that the differential change dv_x can be written as

$$dv_x = \frac{\partial v_x}{\partial x}dx + \frac{\partial v_x}{\partial y}dy + \frac{\partial v_x}{\partial z}dz$$

and similar results can be written for the changes dv_y and dv_z. It follows that the differential change in the vector velocity, $d\boldsymbol{v}$, is given by

$$d\boldsymbol{v} = \boldsymbol{i}\,dv_x + \boldsymbol{j}\,dv_x + \boldsymbol{k}\,dv_z$$

$$= \boldsymbol{i}\left(\frac{\partial v_x}{\partial x}dx + \frac{\partial v_x}{\partial y}dy + \frac{\partial v_x}{\partial z}dz\right) + \boldsymbol{j}\left(\frac{\partial v_y}{\partial x}dx + \frac{\partial v_y}{\partial y}dy + \frac{\partial v_y}{\partial z}dz\right) + \boldsymbol{k}\left(\frac{\partial v_z}{\partial x}dx + \frac{\partial v_z}{\partial y}dy + \frac{\partial v_z}{\partial z}dz\right)$$

$$= \left(\boldsymbol{i}\frac{\partial v_x}{\partial x} + \boldsymbol{j}\frac{\partial v_y}{\partial x} + \boldsymbol{k}\frac{\partial v_z}{\partial x}\right)dx + \left(\boldsymbol{i}\frac{\partial v_x}{\partial y} + \boldsymbol{j}\frac{\partial v_y}{\partial y} + \boldsymbol{k}\frac{\partial v_z}{\partial y}\right)dy + \left(\boldsymbol{i}\frac{\partial v_x}{\partial z} + \boldsymbol{j}\frac{\partial v_y}{\partial z} + \boldsymbol{k}\frac{\partial v_z}{\partial z}\right)dz$$

$$= \frac{\partial \boldsymbol{v}}{\partial x}dx + \frac{\partial \boldsymbol{v}}{\partial y}dy + \frac{\partial \boldsymbol{v}}{\partial z}dz = \nabla\boldsymbol{v} \bullet d\boldsymbol{x}$$

Thus, the relative velocity of a point a distance $d\boldsymbol{x}$ from any given location is given by the dot product of the tensor $\nabla \boldsymbol{v}$ and the differential line element $d\boldsymbol{x}$. This tensor can be written as follows.

$$\nabla \boldsymbol{v} = \begin{pmatrix} \frac{\partial v_x}{\partial x} & \frac{\partial v_x}{\partial y} & \frac{\partial v_x}{\partial z} \\ \frac{\partial v_y}{\partial x} & \frac{\partial v_y}{\partial y} & \frac{\partial v_y}{\partial z} \\ \frac{\partial v_z}{\partial x} & \frac{\partial v_z}{\partial y} & \frac{\partial v_z}{\partial z} \end{pmatrix}$$

Any tensor can be written as the sum of a symmetric and an antisymmetric tensor. Let us do this with the velocity gradient tensor, writing it as

$$\nabla \boldsymbol{v} = \boldsymbol{E} + \boldsymbol{\Omega}$$

where the (symmetric) rate of strain or rate of deformation tensor $\boldsymbol{E}$ is given by

$$\boldsymbol{E} = \frac{1}{2}\left(\nabla \boldsymbol{v} + \nabla \boldsymbol{v}^{\boldsymbol{T}}\right)$$

and the (antisymmetric) vorticity tensor $\boldsymbol{\Omega}$ is given by

$$\boldsymbol{\Omega} = \frac{1}{2}\left(\nabla \boldsymbol{v} - \nabla \boldsymbol{v}^{\boldsymbol{T}}\right)$$

The action of each of these contributions to the velocity gradient will be explored in detail next. First, we consider the vorticity tensor.

Vorticity Tensor $\boldsymbol{\Omega}$

The vorticity tensor $\boldsymbol{\Omega}$ is a skew-symmetric tensor. We can write its components in terms of the components of the velocity gradient as follows.

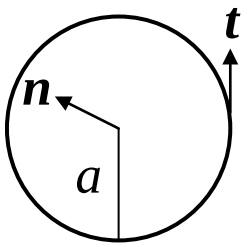
$$\boldsymbol{\Omega} = \begin{pmatrix} 0 & \frac{1}{2}\left(\frac{\partial v_x}{\partial y} - \frac{\partial v_y}{\partial x}\right) & \frac{1}{2}\left(\frac{\partial v_x}{\partial z} - \frac{\partial v_z}{\partial x}\right) \\ \frac{1}{2}\left(\frac{\partial v_y}{\partial x} - \frac{\partial v_x}{\partial y}\right) & 0 & \frac{1}{2}\left(\frac{\partial v_y}{\partial z} - \frac{\partial v_z}{\partial y}\right) \\ \frac{1}{2}\left(\frac{\partial v_z}{\partial x} - \frac{\partial v_x}{\partial z}\right) & \frac{1}{2}\left(\frac{\partial v_z}{\partial y} - \frac{\partial v_y}{\partial z}\right) & 0 \end{pmatrix}$$

A skew-symmetric tensor A_{ij} can be formed from a vector a_k by writing $A_{ij} = \varepsilon_{ijk} a_k$. The vector associated with the vorticity tensor in this manner is $-\frac{1}{2}\boldsymbol{\omega}$, where $\boldsymbol{\omega} = \nabla \times \boldsymbol{v}$ is known as the vorticity vector. Using the relationship between $\boldsymbol{\Omega}$ and $-\frac{1}{2}\boldsymbol{\omega}$, we obtain

$\Omega_{ij} dx_j = -\frac{1}{2}\varepsilon_{ijk} dx_j \omega_k$ or in Gibbs notation, $\boldsymbol{\Omega} \bullet \boldsymbol{dx} = -\frac{1}{2}\boldsymbol{dx} \times \boldsymbol{\omega} = \frac{1}{2}\boldsymbol{\omega} \times \boldsymbol{dx}$

This means that the relative motion that is contributed by the vorticity tensor at a point an infinitesimal distance away from a reference point in a fluid is that caused by a rigid rotation with an angular velocity equal to $\frac{1}{2}\boldsymbol{\omega}$.

Because a fluid does not usually rotate as a rigid body in the manner that a solid does, we should interpret the above statement as implying that the average angular velocity of a fluid element located at a point is one-half the vorticity vector at that point (2). To prove this claim, consider a surface formed by an infinitesimal circle of radius a located at a point $\boldsymbol{x}$. Let the unit normal vector to the surface (perpendicular to the plane of the paper) be $\boldsymbol{n}$, and the unit tangent vector to the circle at any point be $\boldsymbol{t}$.



Apply Stokes's theorem to the velocity field in this circle.

$$\int_S (\nabla \times \boldsymbol{v}) \bullet \boldsymbol{n}\, dS = \oint_C \boldsymbol{v} \bullet \boldsymbol{t}\, ds$$

Here, *dS* is an area element on the surface of the circle S and *ds* is a line element along the circle *C*. Because $\boldsymbol{v} \bullet \boldsymbol{t}$ is the component of the velocity along the periphery of the circle, we can write the average linear velocity along the circle as $\frac{1}{2\pi a}\oint_C \boldsymbol{v} \bullet \boldsymbol{t}\, ds$ and therefore the average angular velocity as $\frac{1}{2\pi a^2}\oint_C \boldsymbol{v} \bullet \boldsymbol{t}\, ds$. From Stokes's theorem, we see that this is equal to the average value of $\left(\frac{1}{2}\nabla \times \boldsymbol{v}\right) \bullet \boldsymbol{n}$ over the surface of the circle. Thus, in the limit as the radius of the circle approaches zero, we find that the average angular velocity around the circle approaches the value of one-half the component of the vorticity vector in a direction perpendicular to the surface of the circle. We also can show (see Batchelor, page 82) that the angular momentum of a spherical

element of fluid is equal to one-half the vorticity times the moment of inertia of the fluid, just as it is for a rigid body.

Vorticity Vector

The vorticity $\boldsymbol{\omega} = \nabla \times \boldsymbol{v}$, is an important entity in fluid mechanics. It is transported from one place to another in a fluid by convective and molecular means, just as energy and species are, and an appropriate partial differential equation that governs its transport can be written. In addition, vorticity also is intensified by the stretching of vortex lines, a mechanism that is not present in the transport of energy and species. One reason for working with the equations of vorticity transport is that pressure is absent as a dependent variable in those equations. It can be shown that if a fluid mass begins with zero vorticity, and the fluid is inviscid (meaning the viscosity is zero), the vorticity will remain zero in that fluid mass. A flow in which the vorticity is zero is known as an irrotational flow.

Vorticity is generated at fluid-solid interfaces and at fluid-fluid interfaces. Vorticity cannot be generated internally within an incompressible fluid. This is the reason why, in a high Reynolds number flow (implying weak viscous effects) past a rigid body, most of the flow can be described by using the equations that apply to irrotational flow, with the vorticity being confined to a boundary layer near the surface of the body.

Vortex Lines and Tubes

Just as a streamline is a curve to which the velocity vector is tangent everywhere, we can define a vortex line as a curve to which the vorticity is tangent everywhere. If the components of the vorticity $\boldsymbol{\omega} = \nabla \times \boldsymbol{v}$ are $\left(\omega_x, \omega_y, \omega_z\right)$, then we can write the equations of the space curves that are vortex lines as

$$\frac{dx}{\omega_x} = \frac{dy}{\omega_y} = \frac{dz}{\omega_z}.$$

The surface that is formed by all the vortex lines passing through a closed reducible curve is known as a vortex tube. If we construct an open surface *S* bounded by this closed curve *C*, we can define the strength of the vortex tube as $\int_S \boldsymbol{dS} \bullet \boldsymbol{\omega}$. By using Stokes's theorem, we can see that this is the circulation $\oint_C \boldsymbol{v} \bullet \boldsymbol{t}\, ds$ where C is any closed curve around the vortex tube, $\boldsymbol{t}$ is a unit tangent vector to the curve at any point, and ds is a line element.

Rate of Strain or Rate of Deformation Tensor $\boldsymbol{E}$

From the above discussion of the vorticity tensor, you can see that the role of that tensor is to describe the instantaneous angular velocity of a fluid element, but that it contributes nothing to deformation of elements. Now, we move on to discuss the significance of the rate of strain tensor, which contains all the information about the deformation.

The rate of strain tensor is a symmetric tensor. We can write its components in terms of the components of the velocity gradient as follows.

$$\mathbf{E} = \begin{pmatrix} \dfrac{\partial v_x}{\partial x} & \dfrac{1}{2}\left(\dfrac{\partial v_x}{\partial y} + \dfrac{\partial v_y}{\partial x}\right) & \dfrac{1}{2}\left(\dfrac{\partial v_x}{\partial z} + \dfrac{\partial v_z}{\partial x}\right) \\ \dfrac{1}{2}\left(\dfrac{\partial v_y}{\partial x} + \dfrac{\partial v_x}{\partial y}\right) & \dfrac{\partial v_y}{\partial y} & \dfrac{1}{2}\left(\dfrac{\partial v_y}{\partial z} + \dfrac{\partial v_z}{\partial y}\right) \\ \dfrac{1}{2}\left(\dfrac{\partial v_z}{\partial x} + \dfrac{\partial v_x}{\partial z}\right) & \dfrac{1}{2}\left(\dfrac{\partial v_z}{\partial y} + \dfrac{\partial v_y}{\partial z}\right) & \dfrac{\partial v_z}{\partial z} \end{pmatrix}$$

The diagonal elements of E

Following Aris (1) closely, consider a line element $d\boldsymbol{x}$ with a length ds.

$$\frac{d}{dt}\left(ds^2\right) = \frac{d}{dt}\left(d\boldsymbol{x} \bullet d\boldsymbol{x}\right)$$

Using the fact that $\dfrac{d\boldsymbol{x}}{dt} = d\boldsymbol{v}$, the above result can be rewritten as

$$2\, ds\, \frac{d}{dt}\left(ds\right) = 2\, d\boldsymbol{x} \bullet d\boldsymbol{v} = 2\, d\boldsymbol{x} \bullet \left(\nabla \boldsymbol{v} \bullet d\boldsymbol{x}\right) = 2\, d\boldsymbol{x} \bullet \boldsymbol{E} \bullet d\boldsymbol{x} + 2\, d\boldsymbol{x} \bullet \boldsymbol{\Omega} \bullet d\boldsymbol{x}$$

The second term in the far-right-side is zero because $\boldsymbol{\Omega}$ is an antisymmetric tensor. To see this, we write

$d\boldsymbol{x} \bullet \boldsymbol{\Omega} \bullet d\boldsymbol{x} = \Omega_{ij} dx_i dx_j \;=\; \Omega_{ji} dx_j dx_i \;=\; -\Omega_{ij} dx_i dx_j$ so that $\Omega_{ij} dx_i dx_j = 0$.

In the above result, after writing the result in index notation, we first exchange the indices i and j to obtain an intermediate result, and then use the antisymmetry property to write $\Omega_{ij} = -\Omega_{ji}$.

Therefore, we find that

$ds\, \dfrac{d}{dt}\left(ds\right) = d\boldsymbol{x} \bullet \boldsymbol{E} \bullet d\boldsymbol{x}$ from which, by dividing through by ds^2 we can write

$$\frac{1}{ds}\frac{d}{dt}(ds) = \frac{d\boldsymbol{x}}{ds} \bullet \boldsymbol{E} \bullet \frac{d\boldsymbol{x}}{ds}$$

The vector $d\boldsymbol{x} / ds$ is a unit vector pointing in the direction of the infinitesimal vector $d\boldsymbol{x}$. Therefore, we can think of the right side of the above result as the "double projection" of the tensor $\boldsymbol{E}$ in that direction. The term "projection" is used in a loose sense here. The physical meaning is clear. The rate of strain of a line element pointing in any direction at a given point (which is the time rate of change of length, divided by the length) is the dot product of a unit vector in that direction with the dot product of the rate of strain tensor with the same unit vector. Let us choose the direction to be the $x-$direction. In this case, the rate of strain of a line element in that direction is simply E_{11}, which is equal to $\frac{\partial v_x}{\partial x}$. In a like manner, the rate of strain of a line element in the $y-$direction is $\frac{\partial v_y}{\partial y}$, and that in the $z-$direction is $\frac{\partial v_z}{\partial z}$. This is the physical interpretation of the diagonal elements of the rate of strain tensor.

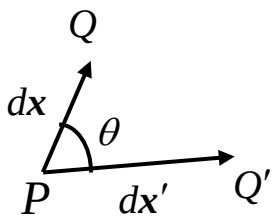
The sum of the diagonal elements of $\boldsymbol{E}$, known as the trace of $\boldsymbol{E}$ is $\nabla \bullet \boldsymbol{v}$. This is known as the rate of dilatation of a fluid element at the given location. To see why, consider a material body occupying a volume V enclosed by the surface S. Let us inquire how V changes with time. We can write the rate of change of the volume of a material body with time as the integral of $d\boldsymbol{S} \bullet \boldsymbol{v}$ over the surface.

$$\frac{dV}{dt} = \int_S d\boldsymbol{S} \bullet \boldsymbol{v} = \int_V \nabla \bullet \boldsymbol{v}\, dV \quad \text{by the divergence theorem.}$$

From the above, we can see that $\lim_{V\to 0} \frac{1}{V}\frac{dV}{dt} = \lim_{V\to 0} \frac{1}{V}\int_V \nabla \bullet \boldsymbol{v}\, dV = \nabla \bullet \boldsymbol{v}$. So, the trace of $\boldsymbol{E}$ is the rate of increase in the volume of an infinitesimal element, divided by its volume, and is called the rate of dilatation. When the flow is incompressible, the rate of dilatation is zero.

The off-diagonal elements of $\boldsymbol{E}$

Now, consider two line elements $d\boldsymbol{x}$ and $d\boldsymbol{x}'$ at a given point $\boldsymbol{x}$ and let the angle between them be θ.



Let us investigate the time rate of change of the dot product of the vectors $d\boldsymbol{x}$ and $d\boldsymbol{x}'$.

$$\frac{d}{dt}\left(ds\, ds'\cos\theta\right) = \frac{d}{dt}\left(d\boldsymbol{x}\bullet d\boldsymbol{x}'\right) = dv_i dx_i' + dx_i dv_i'$$
$$= \frac{\partial v_i}{\partial x_j} dx_j dx_i' + \frac{\partial v_i}{\partial x_j} dx_j' dx_i$$

In writing the result in the second term in the second line, we have used the fact that the infinitesimal change dv_i' is the change in the velocity over an infinitesimal distance in the direction of the vector $d\boldsymbol{x}'$. Interchanging the indices i and j in that second term permits us to combine the two terms.

$$\frac{d}{dt}\left(ds\, ds'\cos\theta\right) = \left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\right) dx_i' dx_j = 2\, E_{ij}\, dx_i' dx_j$$

Dividing both sides by $ds\, ds'$ yields

$$\frac{1}{ds\, ds'}\frac{d}{dt}\left(ds\, ds'\cos\theta\right) \;=\; 2\, E_{ij}\, \frac{dx_i'}{ds'}\frac{dx_j}{ds} = 2\,\frac{d\boldsymbol{x}'}{ds'}\bullet\boldsymbol{E}\bullet\frac{d\boldsymbol{x}}{ds}$$

So, we see that if we take the dot product of $\boldsymbol{E}$ with unit vectors in two different directions in succession (the order is immaterial because $\boldsymbol{E}$ is symmetric), the result is the left side of the above equation. Let us work out the differentiation in the left side.

$$\frac{1}{ds\, ds'}\frac{d}{dt}\left(ds\, ds'\cos\theta\right) = \cos\theta\left[\frac{1}{ds}\frac{d}{dt}\left(ds\right) + \frac{1}{ds'}\frac{d}{dt}\left(ds'\right)\right] - \sin\theta\frac{d\theta}{dt}$$

The term in square brackets in the right side is the sum of the individual rates of strain of the two line elements. We can see that the above result reduces to the earlier result we obtained when the two vectors dx_i and dx_i' are the same. Let us consider the case when the two vectors are orthogonal to each other. In this case, we obtain

$$\frac{d\boldsymbol{x}'}{ds'}\bullet\boldsymbol{E}\bullet\frac{d\boldsymbol{x}}{ds} = \;-\frac{1}{2}\frac{d\theta}{dt}$$

So, the sequential dot products of $\boldsymbol{E}$ with unit vectors in two orthogonal directions yields one-half the rate of decrease of the angle between the unit vectors in those directions. If we choose these two orthogonal directions to coincide with any two coordinate directions, then the dot products yield the off-diagonal elements of $\boldsymbol{E}$. For example, if we use x and $y-$ directions, the element is $E_{12}\left(=E_{21}\right)$. Similar physical interpretations can be given to the other off-diagonal elements of the rate of strain tensor. Thus, the off-diagonal elements describe shear deformation of the fluid.

There are three mutually orthogonal directions associated with the symmetric tensor $\boldsymbol{E}$ that are known as its eigenvector or principal directions. We can use a basis set built from these principal

directions to describe the components of the tensor. If we do, the tensor will be diagonal. The off-diagonal elements will be zero, so that the rate of change of the angles between the principal directions is zero; of course the entire set of principal axes can rotate, and in fact it does, with the angular velocity $\frac{1}{2}\boldsymbol{\omega} = \frac{1}{2}\nabla \times \boldsymbol{v}$.

Instantaneous Deformation of a Fluid Element

Based on all of the above material on kinematics, we can conclude that in a flow, an infinitesimal spherical element of fluid undergoes translation, rotation, and deformation in general. It deforms into an ellipsoid whose axes are aligned with the principal axes of the rate of strain tensor. This ellipsoid also rotates with an instantaneous angular velocity that is equal to the one-half of the vorticity of the fluid at the given point.

Some good sources for further study are listed below.

References

1. R. Aris, Vectors, Tensors, and the Basic Equations of Fluid Mechanics, Dover Publications, 1989, Chapter 4 (original by Prentice-Hall, 1962).

2. G.K. Batchelor, An Introduction to Fluid Dynamics, Cambridge University Press, 1967, Chapter 5.

3. C. Truesdell, Kinematics of Vorticity, Indiana University Press, 1954.

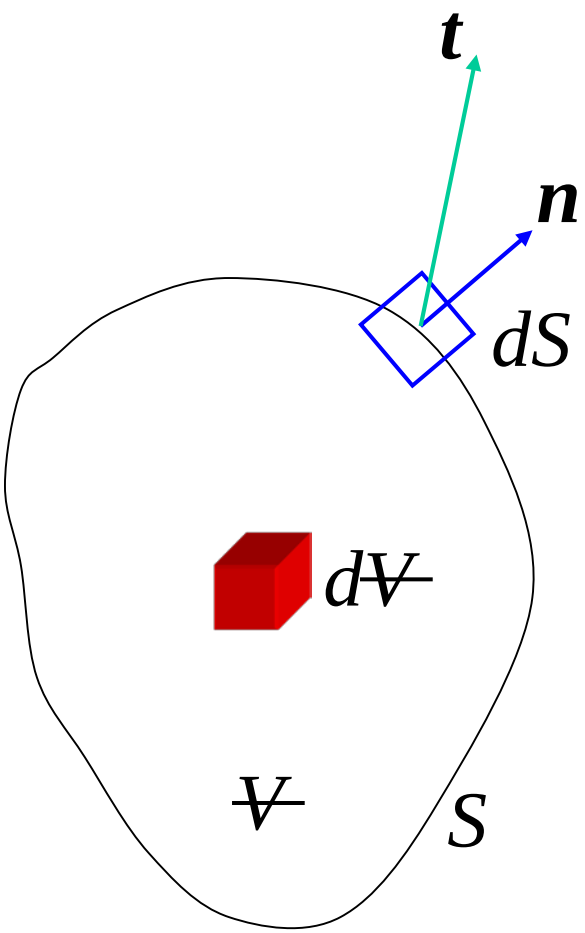
4. J. Serrin, Mathematical Principles of Classical Fluid Dynamics, Handbuch der Physik VIII/1, Ed. S. Flugge, 1959.

5. P.G. Saffman, Vortex Dynamics, Cambridge University Press, 1992.

Navier-Stokes Equation: Principle of Conservation of Momentum

Newton formulated the principle of conservation of momentum for rigid bodies. It took some time for the corresponding version for a continuum, representing a fluid, to be developed. The result is attributed to Cauchy, and is known as Cauchy's equation (1). A derivation of Cauchy's equation is given first. Then, by using a Newtonian constitutive equation to relate stress to rate of strain, the Navier-Stokes equation is derived.

The principle of conservation of momentum is applied to a fixed volume of arbitrary shape in space that contains fluid. This volume is called a "Control Volume." Fluid is permitted to enter or leave the control volume. A control volume $\forall$ is shown in the sketch.



Also marked on the sketch is the bounding surface S of this control volume, called the control surface; an element of surface area dS and the unit outward normal (vector) to that area element, $\boldsymbol{n}$ are shown as well. In addition, the vector stress $\boldsymbol{t}$ exerted by the neighboring fluid on the fluid in the control volume on the area element dS also is displayed. The stress vector will be discussed in more detail shortly. We note that the vector symbol $\boldsymbol{dS} = \boldsymbol{n}\, dS$ is used to represent a directed differential area element on the surface.

We begin with a verbal statement of the principle of conservation of momentum.

Rate of increase of momentum of material within the control volume = Net rate at which momentum enters the control volume with the flowing fluid + Sum of the forces acting on the fluid in the control volume

Next, we shall develop a mathematical representation of the above statement. Because the density ρ of the fluid and the velocity $\boldsymbol{v}$ can both depend on position, to determine the rate of increase of momentum in the control volume, first it is necessary to determine the mass content in the differential volume element $d\forall$. Multiplying the differential volume by the density at that location gives the amount of mass in the differential volume element as $\rho\, d\forall$, and multiplying this mass by the velocity at that location gives the momentum content of the fluid in the differential volume element. The total momentum in the control volume $\forall$ is obtained by integrating this product over the entire control volume. Therefore,

Momentum content of the fluid in the control volume $= \int_{\forall} \rho \boldsymbol{v}\, d\forall$

The time rate of change of this momentum content of the fluid in the control volume is obtained by differentiating the above result with respect to time. Because the control volume is fixed in space, the time derivative can be taken inside the integral and becomes a partial derivative in time, obtained while keeping spatial coordinates fixed. Thus, the left side in the verbal statement of the principle of conservation of momentum is $\int_{\forall} \frac{\partial}{\partial t}(\rho \boldsymbol{v})\, d\forall$, where t represents time.

Next, we need to develop a result for the net rate of entry of momentum with the fluid flowing into and out of the control volume through the control surface. For this, we need to consider the differential area element dS. The rate of influx of mass into the control volume through this differential area is given by $-\rho(\boldsymbol{v} \bullet \boldsymbol{n})\, dS$. Therefore, the rate of influx of momentum with the fluid flowing into the control volume through the area element is $-\rho \boldsymbol{v}(\boldsymbol{v} \bullet \boldsymbol{n})\, dS$. Integrating this result over the entire control surface leads to the following result.

Net rate of influx of momentum into the control volume with the flow =

$$-\int_{S} \rho \boldsymbol{v}(\boldsymbol{v} \bullet \boldsymbol{n})\, dS = -\int_{S} (\rho \boldsymbol{v}\boldsymbol{v}) \bullet \boldsymbol{dS}$$

Using the divergence theorem, the right side of the above equation becomes $-\int_{\forall} \nabla \bullet (\rho \boldsymbol{v}\boldsymbol{v})\, d\forall$.

Now, we turn our attention to summing the forces acting on the fluid in the control volume. These can be broadly divided into body forces and contact forces. Body forces act on every volume element within the fluid occupying the control volume at any given instant, and do not need contact with the material to exert their influence. The most common example of a body force is the gravitational force on objects. Other examples include electrical or magnetic forces. Let us use the symbol $\boldsymbol{f}$ to designate the body force per unit mass acting on the fluid in the control volume. In the case of the gravitational force exerted by Earth on objects, $\boldsymbol{f} = \boldsymbol{g}$, where $\boldsymbol{g}$ is the

acceleration due to gravity vector, pointed toward the center of mass of Earth. It is straightforward to work out the result for the body force on the fluid in the control volume. Once again, we begin with the differential volume element $d\forall$. The mass of fluid in this volume element is $\rho\, d\forall$ so that the body force on this mass is $\rho \boldsymbol{f}\, d\forall$. Adding up all the contributions from such volume elements in the control volume we obtain the following result.

$$\text{Body force acting on the fluid in the control volume} = \int_{\forall} \rho \boldsymbol{f}\, d\forall$$

The contact force on the fluid located at the surface of the control volume arises from intermolecular forces exerted by the molecules on the outside of the surface on the molecules on the inside of the surface. This force, expressed as a result per unit area, is termed the stress $\boldsymbol{t}$. The magnitude and direction of this vector will, in general, depend on the location, as well as the orientation of the area element, given by the direction of the unit normal $\boldsymbol{n}$. Multiplying the stress $\boldsymbol{t}$ by the area dS yields a contact force on the area element that is equal to $\boldsymbol{t}\, dS$. Adding up all the contributions over the surface of the control volume leads to the following result for the total contact force acting on the fluid in the control volume.

$$\text{Contact force acting on the fluid in the control volume} = \int_{S} \boldsymbol{t}\, dS$$

Therefore, the principle of conservation of momentum can be cast in mathematical form as follows:

$$\int_{\forall} \frac{\partial}{\partial t}(\rho \boldsymbol{v})\, d\forall = -\int_{\forall} \nabla \bullet (\rho \boldsymbol{v}\boldsymbol{v})\, d\forall + \int_{\forall} \rho \boldsymbol{f}\, d\forall + \int_{S} \boldsymbol{t}\, dS$$

Now, consider shrinking $\forall$ to zero, and achieving this by permitting a characteristic length scale ℓ to approach zero. The volume integrals will approach zero proportional to ℓ^3, whereas the surface area will approach zero proportional to ℓ^2. Therefore, if we divide through by the surface area, we can write

$$\lim_{\ell \to 0} \frac{1}{S}\left[\int_{\forall} \left(\frac{\partial}{\partial t}(\rho \boldsymbol{v}) + \nabla \bullet (\rho \boldsymbol{v}\boldsymbol{v}) - \rho \boldsymbol{f} \right) d\forall \right] = 0$$

Hence, we can conclude that

$$\lim_{\ell \to 0}\left[\frac{1}{S} \int_{S} \boldsymbol{t}\, dS \right] = 0$$

This implies that the stresses on a fluid are in local equilibrium. This result can be used to establish that the stress vector on an area element at any location in a fluid is given by the equation $\boldsymbol{t} = \boldsymbol{n} \bullet \boldsymbol{T}$, where $\boldsymbol{T}$ is a second order tensor, called the stress tensor in the fluid at that point. For details regarding how this can be established, you can consult pages 99-101 of Aris (1). Furthermore, by invoking the principle of conservation of angular momentum, it can be shown that except in rare cases that need not concern us, the stress tensor is symmetric. Substituting this new result for the stress vector allows us to rewrite the result for the total contact force acting on the fluid in the

control volume as $\int_S \boldsymbol{n} \bullet \boldsymbol{T}\, dS$, or $\int_S \boldsymbol{dS} \bullet \boldsymbol{T}$. Upon using the divergence theorem, this can be shown to be equal to the volume integral $\int_{\not V} \nabla \bullet \boldsymbol{T}\, d\not V$.

Using the results developed above, we can write the following mathematical statement representing the principle of conservation of momentum applied to a control volume.

$$\int_{\not V} \frac{\partial}{\partial t}(\rho \boldsymbol{v})\, d\not V = -\int_{\not V} \nabla \bullet (\rho \boldsymbol{v}\boldsymbol{v})\, d\not V + \int_{\not V} \rho \boldsymbol{f}\, d\not V + \int_{\not V} \nabla \bullet \boldsymbol{T}\, d\not V$$

This result can be rewritten as follows.

$$\int_{\not V} \left[\frac{\partial}{\partial t}(\rho \boldsymbol{v}) + \nabla \bullet (\rho \boldsymbol{v}\boldsymbol{v}) - \rho \boldsymbol{f} - \nabla \bullet \boldsymbol{T} \right] d\not V = \boldsymbol{0}$$

We wish to conclude that at every point in the fluid, the integrand of the above result must be zero. To do this we can use the same arguments employed in deriving the equation of conservation of mass. That is, the control volume is arbitrary, and the integrand is a continuous vector function of position. Thus, with each component of the vector function, we can argue that if it is non-zero at any location in the fluid, then the control volume can be selected as the neighborhood of that point in which it retains the same sign, leading to a violation of the above integral balance. Therefore, the integrand must be zero at every point in in the fluid.

$$\frac{\partial}{\partial t}(\rho \boldsymbol{v}) + \nabla \bullet (\rho \boldsymbol{v}\boldsymbol{v}) - \rho \boldsymbol{f} - \nabla \bullet \boldsymbol{T} = \boldsymbol{0}$$

or

$$\boxed{\frac{\partial}{\partial t}(\rho \boldsymbol{v}) + \nabla \bullet (\rho \boldsymbol{v}\boldsymbol{v}) = \rho \boldsymbol{f} + \nabla \bullet \boldsymbol{T}}$$

This is known as Cauchy's equation. We can combine the two terms in the left side. Using index notation for convenience,

$$\frac{\partial}{\partial t}(\rho \boldsymbol{v}) + \nabla \bullet (\rho \boldsymbol{v}\boldsymbol{v}) = \frac{\partial}{\partial t}(\rho v_i) + \frac{\partial}{\partial x_j}(\rho v_j v_i)$$

$$= v_i \frac{\partial \rho}{\partial t} + \rho \frac{\partial v_i}{\partial t} + v_i \frac{\partial}{\partial x_j}(\rho v_j) + \rho v_j \frac{\partial v_i}{\partial x_j}$$

$$= v_i \left[\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho v_j) \right] + \rho \left[\frac{\partial v_i}{\partial t} + v_j \frac{\partial v_i}{\partial x_j} \right]$$

$$= \boldsymbol{v} \left[\cancel{\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \boldsymbol{v})} \right] + \rho \left[\frac{\partial \boldsymbol{v}}{\partial t} + \boldsymbol{v} \bullet \nabla \boldsymbol{v} \right]$$

where the first term in square brackets is zero because of the continuity equation. Thus, we find

$$\frac{\partial}{\partial t}(\rho \boldsymbol{v}) + \nabla \bullet (\rho \boldsymbol{v}\boldsymbol{v}) = \rho \left[\frac{\partial \boldsymbol{v}}{\partial t} + \boldsymbol{v} \bullet \nabla \boldsymbol{v} \right] = \rho \frac{d\boldsymbol{v}}{dt}$$

where $\frac{d\boldsymbol{v}}{dt} = \frac{\partial \boldsymbol{v}}{\partial t} + \boldsymbol{v} \bullet \nabla \boldsymbol{v}$ is the material derivative of the velocity, namely, the derivative with respect to time taken while travelling with the fluid at any given point. Therefore, Cauchy's equation can be rewritten as

$$\boxed{\rho \frac{d\boldsymbol{v}}{dt} = \rho \boldsymbol{f} + \nabla \bullet \boldsymbol{T}}$$

When the body force is Earth's gravity, we can write Cauchy's equation as

$$\boxed{\rho \frac{d\boldsymbol{v}}{dt} = \rho \boldsymbol{g} + \nabla \bullet \boldsymbol{T}}$$

where $\boldsymbol{g}$ was defined earlier as the acceleration due to gravity vector.

In index notation, Cauchy's equation is written as follows.

$$\boxed{\rho \frac{dv_j}{dt} = \rho f_j + \frac{\partial T_{ij}}{\partial x_i}}$$

Navier-Stokes Equation

The continuity (or conservation of mass) equation and Cauchy's equation are insufficient by themselves, because we have too many unknowns. The density and the components of the velocity vector field constitute four unknowns, while the scalar conservation of mass equation and the vector conservation of momentum equation provide four scalar balances. But, we also have six unknown components of the symmetric stress tensor in the conservation of momentum equation. Thus, in order to pose a solvable system of equations, we need to have additional information. There is no other conservation principle we can use. Therefore, we must resort to a "constitutive equation" known as a phenomenological model, to describe the connection between the stress tensor and the components of the velocity vector. It seems plausible to postulate that the stress in a fluid $\boldsymbol{T}$ is related to the velocity gradient $\nabla \boldsymbol{v}$, which can be written as the sum of a symmetric tensor, namely the rate of deformation $\boldsymbol{E}$, and an antisymmetric tensor, namely the vorticity $\boldsymbol{\Omega}$. Most derivations postulate that the stress must only depend upon the rate of deformation $\boldsymbol{E}$, because the vorticity in a fluid $\boldsymbol{\Omega}$ describes only the instantaneous angular motion of an element of fluid, and does not have anything to do with deformation. Others argue that a "principle of material frame indifference" requires that the same constitutive equation apply to a fluid whether one considers a laboratory reference frame or one that is instantaneously rotating at the local angular velocity at a point in the fluid. This argument has been contested in the literature. Aris (1) and Batchelor (2) provide an alternative viewpoint, postulating a general tensorial connection between $\boldsymbol{T}$ and $\nabla \boldsymbol{v}$, and showing that isotropy arguments lead to the exclusion of $\boldsymbol{\Omega}$ from the

relationship, leading to a dependence of the state of stress only on the rate of deformation. In any case, this is where we begin our development, namely, by postulating that $\boldsymbol{T}$ is some function of $\boldsymbol{E}$. As discussed in Aris (1) and Slattery (3), one can take advantage of the fact that both the stress and rate of deformation are symmetric tensors, and write the most general connection between two symmetric tensors in the following form.

$$\boldsymbol{T} = \chi_0 \boldsymbol{I} + \chi_1 \boldsymbol{E} + \chi_2 \boldsymbol{E} : \boldsymbol{E}$$

Here, $\chi_o, \chi_1,$ and χ_2 are functions of the three invariants of the tensor $\boldsymbol{E}$. If you are wondering about higher order terms in what appears to be an expansion in power series, it is shown in Aris (1) that higher powers of $\boldsymbol{E}$ can be reduced to results that depend only on the lower powers included here and the three invariants of $\boldsymbol{E}$.

The task now is to establish the nature of the functions in the relationship between $\boldsymbol{T}$ and $\boldsymbol{E}$. Here, we simplify the problem greatly by assuming that the elements of the stress tensor depend only linearly on the elements of the rate of deformation tensor. This is known as a Newtonian relationship, to honor Newton who first postulated a linear dependence of the shear stress on the velocity gradient in a one-dimensional context. The first invariant of $\boldsymbol{E}$ is the trace of $\boldsymbol{E}$, which we know is the divergence of the velocity field $\nabla \bullet \boldsymbol{v}$. This, of course, is linear in the elements of $\boldsymbol{E}$. The second invariant is quadratic in the elements of $\boldsymbol{E}$, and the third invariant, which is the determinant of $\boldsymbol{E}$, is cubic in the elements of $\boldsymbol{E}$. Thus, neither the second nor the third invariant can appear in the relationship between the stress and the rate of deformation for a Newtonian fluid. Furthermore, we can see that the function χ_2 must be zero because of the quadratic dependence of $\boldsymbol{E} : \boldsymbol{E}$ on the elements of $\boldsymbol{E}$, χ_1 has to be a scalar constant independent of the elements of $\boldsymbol{E}$, and χ_0 can, at best, be a linear function of $\nabla \bullet \boldsymbol{v}$. Using this information, the Newtonian constitutive equation is written as follows.

$$\boldsymbol{T} = \left[-p + \lambda \nabla \bullet \boldsymbol{v} \right] \boldsymbol{I} + 2\mu \boldsymbol{E}$$

Here, p will be seen to be the pressure field in the fluid shortly, and λ and μ are identified as physical properties of a given fluid. The symbol μ stands for a physical property known as the coefficient of shear viscosity, or simply viscosity. Sometimes, it is called dynamic viscosity to distinguish it from another property known as kinematic viscosity. There appears to be no specific name for the property represented by the symbol λ, but there is a name for $\kappa = \lambda + \frac{2}{3}\mu.$ It is called the coefficient of bulk viscosity and is used to account for dissipation that occurs in a fluid that is rapidly expanded and contracted. In incompressible flow, the velocity field is solenoidal so that this type of dissipation does not occur, making the value of the coefficient of bulk viscosity moot. Historically, the so-called Stokes hypothesis assumed that $\kappa = 0$, but this is not necessarily true. In any case, the value of κ will prove to be unimportant for incompressible flow, which implies $\nabla \bullet \boldsymbol{v} = 0$.

We can see that in the absence of motion $\boldsymbol{v} \equiv \boldsymbol{0}$, so that $\boldsymbol{E} \equiv \boldsymbol{0}$ as well. Thus, the result for the stress reduces to

$$\boldsymbol{T} = -p\boldsymbol{I}$$

That is, the state of stress in a stationary fluid is isotropic. The stress vector on any area element at a given location is given by

$$\boldsymbol{t} = \boldsymbol{n} \bullet \boldsymbol{T} = -p\left(\boldsymbol{n} \bullet \boldsymbol{I}\right) = -p\boldsymbol{n}$$

At a given point, regardless of the orientation of the area element, the stress is of the same magnitude p, termed pressure. Pressure is usually positive, so that the stress in a stationary fluid points in the direction opposite to the normal vector to the area element. Thus, a positive pressure results in compressive stress on an element of fluid at any given location in the fluid.

To describe the more general case of a moving fluid, we first need to insert the Newtonian constitutive equation into the Cauchy equation. Thus, we write

$$\nabla \bullet \boldsymbol{T} = \nabla \bullet \left\{\left[-p + \lambda \nabla \bullet \boldsymbol{v}\right]\boldsymbol{I} + 2\mu\, \boldsymbol{E}\right\} = -\nabla p + \nabla\left(\lambda \nabla \bullet \boldsymbol{v}\right) + 2\nabla \bullet \left(\mu\, \boldsymbol{E}\right)$$

The viscosity μ is commonly assumed to be a constant to simplify the governing equation. We find $\nabla \bullet \boldsymbol{E}$ as follows.

$$\nabla \bullet \boldsymbol{E} = \frac{\partial E_{ij}}{\partial x_i} = \frac{1}{2}\left[\frac{\partial}{\partial x_i}\left(\frac{\partial v_i}{\partial x_j}\right) + \frac{\partial}{\partial x_i}\left(\frac{\partial v_j}{\partial x_i}\right)\right]$$
$$= \frac{1}{2}\left[\frac{\partial}{\partial x_j}\left(\frac{\partial v_i}{\partial x_i}\right) + \frac{\partial}{\partial x_i}\left(\frac{\partial v_j}{\partial x_i}\right)\right] = \frac{1}{2}\left[\nabla\left(\nabla \bullet \boldsymbol{v}\right) + \nabla \bullet \left(\nabla \boldsymbol{v}\right)\right]$$

We shall only be concerned with incompressible flow so that we can set $\nabla \bullet \boldsymbol{v} = 0$. Substituting for $\nabla \bullet \boldsymbol{E}$ in the result for $\nabla \bullet \boldsymbol{T}$ and using the assumptions stated here yields

$$\nabla \bullet \boldsymbol{T} = -\nabla p + \mu \nabla \bullet \left(\nabla \boldsymbol{v}\right) = -\nabla p + \mu \nabla^2 \boldsymbol{v}$$

Now, we can return to the Cauchy equation and substitute the above result for $\nabla \bullet \boldsymbol{T}$ in the right side to obtain the following version of the principle of conservation of momentum that we shall use extensively.

$$\boxed{\rho \frac{d\boldsymbol{v}}{dt} = -\nabla p + \rho \boldsymbol{g} + \mu \nabla^2 \boldsymbol{v}}$$ or alternatively

$$\underbrace{\rho\left(\frac{\partial \boldsymbol{v}}{\partial t}+(\boldsymbol{v}\bullet\nabla)\boldsymbol{v}\right)}_{\text{inertia force}}=\underbrace{-\nabla p}_{\substack{\text{pressure}\\ \text{force}}}+\underbrace{\rho\boldsymbol{g}}_{\substack{\text{gravitational}\\ \text{force}}}+\underbrace{\mu\nabla^2\boldsymbol{v}}_{\substack{\text{viscous}\\ \text{force}}}$$

This equation was first derived by Navier using molecular arguments, and later by Stokes for a continuum. Thus, it is known as the Navier-Stokes equation for incompressible flow and constant viscosity. It is a vector balance in which each term has the dimensions of force per unit volume. The left side is the inertia force. The first term in the right side represents the pressure force, the second the gravitational force, and the third, the viscous force. The viscosity is a physical property, and therefore depends on the thermodynamic state of the fluid. It is sensitive to temperature in both gases and liquids, and relatively insensitive to pressure under commonly encountered conditions, with the exception of unusually large pressures. For more information, you can consult Bird et al. (4). Also you will find a good introduction to non-Newtonian constitutive relations in Chapter 8 of Bird et al.

References

1. R. Aris, Vectors, Tensors, and the Basic Equations of Fluid Mechanics, Dover Publications, 1989 (original by Prentice-Hall, 1962).

2. G. K. Batchelor, An Introduction to Fluid Dynamics, Cambridge University Press, 1967.

3. J. C. Slattery, Momentum, Energy, and Mass Transfer in Continua, McGraw-Hill, 1972.

4. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, Transport Phenomena, Wiley, 2007.

Standard procedure for posing and solving a fluid mechanics problem

The equation of continuity and the Navier-Stokes equation are called the "governing equations" in fluid mechanics problems, and form the starting point for posing a variety of such problems. A typical approach for setting up and solving a fluid mechanics problem is outlined below.

1. Choose a natural coordinate system in which the boundaries of the domain can be described in a simple manner. The simplest description of a boundary is that it is a surface at which one of the coordinates is constant. As examples, for flow between parallel plates, we might choose a rectangular coordinate system, for flow in a circular tube, a cylindrical polar coordinate system, and for flow over a sphere, a spherical polar coordinate system.

2. Write the governing equations in component form.

3. Write initial and boundary conditions on the velocity field.

4. If the viscosity (or density) is a function of position through its dependence on pressure, additional equations stating the relationship should be written. If the viscosity (or density) depends on temperature or composition, the appropriate governing equations and initial and boundary conditions for those fields as well as the connection between the viscosity (or density) and temperature or composition should be written.

5. Make physically sound simplifying assumptions and eliminate terms in the equations that are zero, or close to zero.

6. Now, proceed to solve the resulting system of equations for the velocity and pressure fields.

From the solution, we can calculate entities of interest, such as how the velocity varies with position and time, the force exerted on the boundaries of the domain, and average and maximum velocities for flow in confined domains such as a channel. The velocity distribution in a tube or channel, or in flow over an object, is an important entity when solving the associated heat or mass transport problems and is often sought for that reason.

In the next few pages, the component forms of the governing equations in the most common coordinate systems are given.

The Equation of Continuity for incompressible flow

Rectangular coordinates (x, y, z)

$$\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0$$

Cylindrical polar coordinates (r, θ, z)

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

Spherical polar coordinates (r, θ, ϕ)

$$\frac{\partial \rho}{\partial t} + \frac{1}{r^2}\frac{\partial}{\partial r}(\rho r^2 v_r) + \frac{1}{r\sin\theta}\frac{\partial}{\partial \theta}(\rho v_\theta \sin\theta) + \frac{1}{r\sin\theta}\frac{\partial}{\partial \phi}(\rho v_\phi) = 0$$

The Navier-Stokes equations in rectangular coordinates (x, y, z)

(for incompressible Newtonian flow with constant viscosity μ)

$x-$ component

$$\rho\left(\frac{\partial v_x}{\partial t}+v_x\frac{\partial v_x}{\partial x}+v_y\frac{\partial v_x}{\partial y}+v_z\frac{\partial v_x}{\partial z}\right)=-\frac{\partial p}{\partial x}+\rho g_x+\mu\left(\frac{\partial^2 v_x}{\partial x^2}+\frac{\partial^2 v_x}{\partial y^2}+\frac{\partial^2 v_x}{\partial z^2}\right)$$

$y-$ component

$$\rho\left(\frac{\partial v_y}{\partial t}+v_x\frac{\partial v_y}{\partial x}+v_y\frac{\partial v_y}{\partial y}+v_z\frac{\partial v_y}{\partial z}\right)=-\frac{\partial p}{\partial y}+\rho g_y+\mu\left(\frac{\partial^2 v_y}{\partial x^2}+\frac{\partial^2 v_y}{\partial y^2}+\frac{\partial^2 v_y}{\partial z^2}\right)$$

$z-$ component

$$\rho\left(\frac{\partial v_z}{\partial t}+v_x\frac{\partial v_z}{\partial x}+v_y\frac{\partial v_z}{\partial y}+v_z\frac{\partial v_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\rho g_z+\mu\left(\frac{\partial^2 v_z}{\partial x^2}+\frac{\partial^2 v_z}{\partial y^2}+\frac{\partial^2 v_z}{\partial z^2}\right)$$

The Navier-Stokes equations in cylindrical polar coordinates (r,θ,z)

(for incompressible Newtonian flow with constant viscosity μ)

$r-$component

$$\rho\left[\frac{\partial v_r}{\partial t}+v_r\frac{\partial v_r}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta}-\frac{v_\theta^2}{r}+v_z\frac{\partial v_r}{\partial z}\right]=-\frac{\partial p}{\partial r}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}\left(rv_r\right)\right)+\frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2}-\frac{2}{r^2}\frac{\partial v_\theta}{\partial \theta}+\frac{\partial^2 v_r}{\partial z^2}\right]+\rho g_r$$

$\theta-$component

$$\rho\left[\frac{\partial v_\theta}{\partial t}+v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}+\frac{v_r v_\theta}{r}+v_z\frac{\partial v_\theta}{\partial z}\right]=-\frac{1}{r}\frac{\partial p}{\partial \theta}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}\left(rv_\theta\right)\right)+\frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2}+\frac{2}{r^2}\frac{\partial v_r}{\partial \theta}+\frac{\partial^2 v_\theta}{\partial z^2}\right]+\rho g_\theta$$

$z-$ component

$$\rho\left[\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_z}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta}+v_z\frac{\partial v_z}{\partial z}\right]=-\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2}+\frac{\partial^2 v_z}{\partial z^2}\right]+\rho g_z$$

The Navier-Stokes equations in spherical polar coordinates (r,θ,ϕ)

(for incompressible Newtonian flow with constant viscosity μ)

r – component

$$\rho\left(\frac{\partial v_r}{\partial t}+v_r\frac{\partial v_r}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta}+\frac{v_\phi}{r\sin\theta}\frac{\partial v_r}{\partial \phi}-\frac{v_\theta^2+v_\phi^2}{r}\right)=-\frac{\partial p}{\partial r}$$
$$+\mu\left[\frac{1}{r^2}\frac{\partial^2}{\partial r^2}\left(r^2 v_r\right)+\frac{1}{r^2\sin\theta}\frac{\partial}{\partial\theta}\left(\sin\theta\frac{\partial v_r}{\partial\theta}\right)+\frac{1}{r^2\sin^2\theta}\frac{\partial^2 v_r}{\partial\phi^2}\right]+\rho g_r$$

θ – component

$$\rho\left(\frac{\partial v_\theta}{\partial t}+v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}+\frac{v_\phi}{r\sin\theta}\frac{\partial v_\theta}{\partial \phi}+\frac{v_r v_\theta}{r}-\frac{v_\phi^2\cot\theta}{r}\right)=-\frac{1}{r}\frac{\partial p}{\partial\theta}$$
$$+\mu\left[\frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2\frac{\partial v_\theta}{\partial r}\right)+\frac{1}{r^2}\frac{\partial}{\partial\theta}\left(\frac{1}{\sin\theta}\frac{\partial}{\partial\theta}\left(v_\theta\sin\theta\right)\right)+\frac{1}{r^2\sin^2\theta}\frac{\partial^2 v_\theta}{\partial\phi^2}+\frac{2}{r^2}\frac{\partial v_r}{\partial\theta}-\frac{2}{r^2}\frac{\cot\theta}{\sin\theta}\frac{\partial v_\phi}{\partial\phi}\right]+\rho g_\theta$$

ϕ – **c**omponent

$$\rho\left(\frac{\partial v_\phi}{\partial t}+v_r\frac{\partial v_\phi}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\phi}{\partial \theta}+\frac{v_\phi}{r\sin\theta}\frac{\partial v_\phi}{\partial \phi}+\frac{v_r v_\phi}{r}+\frac{v_\theta v_\phi}{r}\cot\theta\right)=-\frac{1}{r\sin\theta}\frac{\partial p}{\partial\phi}$$
$$+\mu\left[\frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2\frac{\partial v_\phi}{\partial r}\right)+\frac{1}{r^2}\frac{\partial}{\partial\theta}\left(\frac{1}{\sin\theta}\frac{\partial}{\partial\theta}\left(v_\phi\sin\theta\right)\right)+\frac{1}{r^2\sin^2\theta}\frac{\partial^2 v_\phi}{\partial\phi^2}+\frac{2}{r^2\sin\theta}\frac{\partial v_r}{\partial\phi}+\frac{2}{r^2}\frac{\cot\theta}{\sin\theta}\frac{\partial v_\theta}{\partial\phi}\right]+\rho g_\phi$$

Boundary Conditions in Fluid Mechanics

The governing equations for the velocity and pressure fields are partial differential equations that are applicable at every point in a fluid that is being modeled as a continuum. When they are integrated in any given situation, we can expect to see arbitrary functions or constants appear in the solution. To evaluate these, we need additional statements about the velocity field and possibly its gradient at the natural boundaries of the flow domain. Such statements are known as *boundary conditions*. Usually, the specification of the pressure at one point in the system suffices to establish the pressure fields so that we shall only discuss boundary conditions on the velocity field here. For a more detailed discussion of various aspects, the reader is encouraged to consult either Leal (1) or Batchelor (2).

Conditions at a rigid boundary

It is convenient for the purpose of discussion to identify two types of boundaries. One is that at the interface between a fluid and a rigid surface. At such a surface, we shall require that the tangential component of the velocity of the fluid be the same as the tangential component of the velocity of the surface, and similarly the normal component of the velocity of the fluid be the same as the normal component of the velocity of the surface. The former is known as the "no slip" boundary condition, and has been found to be successful in describing most practical situations. It was a subject of controversy in the eighteenth and nineteenth centuries, and was finally accepted because predictions based on assuming it were found to be consistent with observations of macroscopic quantities such as the flow rate through a circular capillary under a given pressure drop. If we designate the velocity of the rigid surface as $\boldsymbol{V}$ and that of the fluid as $\boldsymbol{v}$, and select a unit normal vector to the surface pointing into the fluid at a given location as $\boldsymbol{n}$, the no-slip boundary condition can be stated as

$$\boldsymbol{v} - (\boldsymbol{n} \bullet \boldsymbol{v})\,\boldsymbol{n} = \boldsymbol{V} - (\boldsymbol{n} \bullet \boldsymbol{V})\,\boldsymbol{n} \qquad \text{on a rigid surface} \qquad \text{(no slip)}$$

When there is no mass transfer across the boundary, a purely kinematical consequence is that the normal component of the fluid velocity at the boundary must equal the normal component of the velocity of the rigid surface.

$$\boldsymbol{v} \bullet \boldsymbol{n} = \boldsymbol{V} \bullet \boldsymbol{n} \qquad \text{on a rigid surface} \qquad \text{(kinematic condition)}$$

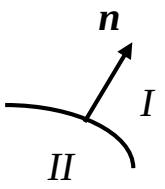
As a consequence of the two conditions, we arrive at the conclusion that the fluid velocity must be equal to the velocity of the rigid surface at every point on it.

$$\boldsymbol{v} = \boldsymbol{V} \qquad \text{on a rigid surface}$$

The no-slip condition has been found to be inapplicable in special circumstances such as at a moving contact line when a drop spreads over a solid surface, or in flow of a rarefied gas through a pore of diameter of the same order of magnitude as the mean free path of the gas molecules. For the types of problems that we shall encounter, it is an adequate boundary condition.

Conditions at a fluid-fluid interface

Sometimes, we encounter a boundary between two fluids. A common example occurs when a liquid film flows down an inclined plane. The surface of the liquid film in contact with the surrounding gas is a fluid-fluid interface. Other examples include the interface between a liquid drop and the surrounding continuous phase or that between two liquid layers. It is convenient to designate the two fluid phases in contact as phase I and phase II.



The unit normal vector $\boldsymbol{n}$ points into phase I here.

It so happens that the velocity fields in fluids I and II are continuous across the interface so long as there is no mass transfer across the interface. This vector condition also can be viewed as being in two parts, one on the continuity of the tangential component of the two velocities, analogous to the no-slip boundary condition at a rigid boundary.

$\boldsymbol{v}_I - (\boldsymbol{n} \bullet \boldsymbol{v}_I)\,\boldsymbol{n} = \boldsymbol{v}_{II} - (\boldsymbol{n} \bullet \boldsymbol{v}_{II})\,\boldsymbol{n}$ at a fluid-fluid interface (continuity of tangential velocity)

If the interface is in motion, we can describe it using the equation $F(t, x, y, z) = 0$ where t is time and F represents some function of time and position represented in Cartesian coordinates here just for convenience. Because $F = 0$ on the interface at all times, the derivative with respect to time following a material particle on the interface, also known as the material derivative, must be zero. That is,

$$\frac{dF}{dt} = \frac{\partial F}{\partial t} + \boldsymbol{v} \bullet \nabla F = 0 \text{ on the interface.}$$

Assuming that F is defined such that ∇F is directed into phase I, the unit normal shown in the sketch is given by $\boldsymbol{n} = \nabla F / |\nabla F|$ so that $\boldsymbol{v} \bullet \nabla F = \boldsymbol{v} \bullet \boldsymbol{n} |\nabla F|$ where the velocity $\boldsymbol{v}$ is that of the interface. In the absence of mass transfer, the normal velocity is continuous across the interface and equal to the velocity of the interface normal to itself, as a kinematical consequence. Thus, the boundary condition on the normal velocity may be stated as follows.

$$\boxed{\boldsymbol{v}_I \bullet \boldsymbol{n} = \boldsymbol{v}_{II} \bullet \boldsymbol{n} = -\frac{1}{|\nabla F|}\,\frac{\partial F}{\partial t}} \text{ at a fluid-fluid interface} \qquad \text{(kinematic condition)}$$

If the interface is fixed in space, F is independent of time so that $\frac{\partial F}{\partial t} = 0$. In this case the above result simplifies to

$$\boxed{\boldsymbol{v_I} \bullet \boldsymbol{n} = \boldsymbol{v_{II}} \bullet \boldsymbol{n} = 0} \text{ at a fluid-fluid interface}$$

Notice that we have two unknown vector fields $\boldsymbol{v}_I$ and $\boldsymbol{v}_{II}$ now, and therefore need twice as many boundary conditions. Therefore, it is not sufficient to write just the above no-slip and kinematic conditions at a fluid-fluid interface. We also need to write a boundary condition connecting the state of stress in each fluid at the interface. This boundary condition is obtained from the principle that the forces on an element of interfacial area of arbitrary shape and size must be in equilibrium because the interface is assumed to have zero thickness and therefore zero mass. The following derivation is based on the development given by Leal (1).

Let us use the symbol A to designate the area of an interfacial element of some arbitrary shape and size, and C to designate the closed curve that forms its boundary. Let the local normal to C be labeled $\boldsymbol{n}_C$. This normal vector to C lies on the tangent plane to the surface at each point. Then, the condition that the total force on the area element must be zero can be written as follows.

$$\int_A \boldsymbol{n} \bullet \left[\boldsymbol{T}_I - \boldsymbol{T}_{II}\right] dA + \int_C \sigma\, \boldsymbol{n}_C\, ds = \boldsymbol{0}$$

In the above equation, the symbols $\boldsymbol{T_I}$ and $\boldsymbol{T_{II}}$ represent the stress tensor in each fluid, $\boldsymbol{n}$ is the unit normal pointing into fluid I, dA is a differential area element in A, σ is the interfacial tension, which can depend on position, and ds is a differential arc length on the closed curve C. Using a version of Stokes theorem, the line integral can be converted to an area integral, permitting us to rewrite the above balance as

$$\int_A \left(\boldsymbol{n} \bullet \left[\boldsymbol{T}_I - \boldsymbol{T}_{II}\right] + \nabla_s \sigma - \sigma\, \boldsymbol{n} \left\{\nabla \bullet \boldsymbol{n}\right\}\right) dA = \boldsymbol{0}$$

In this result, ∇_s is the surface gradient operator which can be written as $\nabla - \boldsymbol{n}\left(\boldsymbol{n} \bullet \nabla\right)$. That is, we remove the part of the gradient vector that is normal to the surface. Because the integrand is a continuous function of position on the interface and the interfacial element chosen is of arbitrary size and shape, we can show that the integrand must be zero at every point on the interface. If any component of the vector integrand is non-zero at any point, then there must exist a neighborhood of that point on the interface in which that component of the integrand would be of the same sign, and the integration could be performed over that neighborhood to yield a non-zero result for that component of the force, thus violating the above equation. Therefore, we obtain the following result, which must be satisfied at every point on a fluid-fluid interface.

$$\boldsymbol{n} \bullet \left[\boldsymbol{T}_I - \boldsymbol{T}_{II}\right] = \sigma\, \boldsymbol{n} \left(\nabla \bullet \boldsymbol{n}\right) - \nabla_s \sigma$$

The divergence of the unit normal is related to the mean curvature H of the interface.

$$\nabla \bullet \boldsymbol{n} = 2H = \left(\frac{1}{R_1} + \frac{1}{R_2} \right)$$

where R_1 and R_2 are the principal radii of curvature of the interface at a given point. For more details about surface geometry, the reader may wish to consult a book such as that by Weatherburn (3). Using the above result, the following equation, called the jump condition on the stress, at a fluid-fluid interface, can be written.

$$\boldsymbol{n} \bullet \left[\boldsymbol{T}_I - \boldsymbol{T}_{II} \right] = 2H\sigma\boldsymbol{n} - \nabla_s \sigma \quad \text{at a fluid-fluid interface} \qquad \text{(jump condition on the stress)}$$

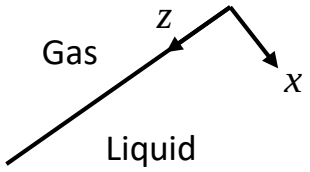
To summarize, in the stress boundary condition, the symbols $\boldsymbol{T}_I$ and $\boldsymbol{T}_{II}$ represent the stress tensor in each fluid, H is the mean curvature of the interface at the point where the condition is being applied, σ is the interfacial tension of the fluid-fluid interface, and ∇_s is the surface gradient operator. The left side in the stress boundary condition is the difference between the stress vectors in fluids I and II at the interface, or the "jump" in stress. This is the reason for the choice of terminology used in describing this condition. The resulting vector is decomposed into a part that is normal to the interface, namely the first term in the right side, and a part that is tangential to the interface, given in the second term in the right side. Sometimes, the condition is written as two separate scalar boundary conditions by writing the tangential and the normal parts separately. In that case, we call the two boundary conditions the "tangential stress balance" and the "normal stress balance."

Tangential Stress Balance

In the types of problems that we shall encounter, the stress boundary condition can be simplified. The interfacial tension at a fluid-fluid interface depends on the temperature and the composition of the interface. If we assume these to be uniform, then the gradient of interfacial tension will vanish everywhere on the interface. This means that the tangential stress is continuous across the interface because the jump in it is zero. Recall that the tangential stress is purely viscous in origin. If τ_t represents this stress component, we can write

$$\tau_{t_I} = \tau_{t_{II}} \qquad \text{at a fluid-fluid interface} \qquad \text{(tangential stress balance)}$$

At a liquid-gas interface, we can further simplify the tangential stress balance. Consider the surface of a liquid film flowing down an inclined plane. Let us assume that the flow is steady and that the film surface is parallel to the inclined plane. In this situation, the normal velocity at the free surface of the liquid is zero in both the liquid and the gas. The sketch depicts the situation.



Because the normal velocity is zero at the free surface, the tangential stress balance simplifies to the following result where the subscripts l and g represent the liquid and gas, respectively.

$$\mu_l \frac{\partial v_{z,l}}{\partial x} = \mu_g \frac{\partial v_{z,g}}{\partial x} \qquad \text{at the free surface}$$

The symbol μ in the above result stands for the dynamic viscosity. If we divide through by the dynamic viscosity of the liquid, we obtain

$$\frac{\partial v_{z,l}}{\partial x} = \frac{\mu_g}{\mu_l} \frac{\partial v_{z,g}}{\partial x} \qquad \text{at the free surface}$$

Because the dynamic viscosity of a gas is small compared with that of a liquid, the right side of the above equation is small, and can be considered negligible. This allows us to write

$$\frac{\partial v_{z,l}}{\partial x} \approx 0 \qquad \text{at the free surface}$$

Sometimes, this condition is represented as that of vanishing shear stress at a free liquid surface. Note that this approximation of the tangential stress condition can be used only when the motivating force for the motion of the liquid is not the motion of the gas. When a gas drags a liquid along, as is the case on a windy day when the wind causes motion in a puddle of liquid, the correct boundary condition equating the tangential stresses must be used.

Normal Stress Balance

The normal stress jump boundary condition actually determines the curvature of the interface at the point in question, and therefore the shape of the entire fluid-fluid interface. This shape is distorted by the flow. Thus, the boundary condition must be applied at an unknown boundary whose shape must be obtained as part of the solution. Fluid mechanical problems involving the application of the normal stress balance at a boundary are complicated, and must be solved numerically unless one assumes the shape distortion to be very small or of a particularly simple form.

For an illustration of a rather simple application of the balance of normal stress, consider a stationary system with no flow, so that $\boldsymbol{v}_I = \boldsymbol{v}_{II} = \boldsymbol{0}$. If we assume the surface tension to be uniform, the jump condition on the stress reduces to just a balance of normal stresses as follows.

$$\boldsymbol{n} \bullet \left[\boldsymbol{T}_I - \boldsymbol{T}_{II} \right] = 2H\sigma \boldsymbol{n}$$

The stress tensor in stationary fluids is simple. $\boldsymbol{T}_I = -p_I \boldsymbol{I}$ and $\boldsymbol{T}_{II} = -p_{II} \boldsymbol{I}$, where p is the pressure, so that the left side becomes $\left(p_{II} - p_I\right)\boldsymbol{n}$. Thus, the normal stress balance reduces to the scalar result

$$p_{II} - p_I = 2H\sigma = \sigma\left(\frac{1}{R_1} + \frac{1}{R_2}\right)$$

which is well-known. Consider a spherical liquid drop or gas bubble that is stationary inside another fluid. For a sphere of radius R, the curvature is uniform, so that $R_1 = R_2 = R$. Neglecting hydrostatic variation of pressure, we can then write

$$\boxed{p_{II} - p_I = \frac{2\sigma}{R}}$$

a result that relates the excess pressure within a spherical bubble or drop to the surface tension and the radius.

In the problems that we shall analyze, we shall always assume the shape of the interface to be the static shape and as being specified. Therefore, we shall not be able to necessarily satisfy the balance of normal stress.

References

1. L.G. Leal, Advanced Transport Phenomena: Fluid Mechanics and Convective Transport Processes, Cambridge University Press, 2007.

2. G. K. Batchelor, An Introduction to Fluid Dynamics, Cambridge University Press, 1967.

3. C.E. Weatherburn, Differential Geometry of Three Dimensions, Cambridge University Press, 1961.

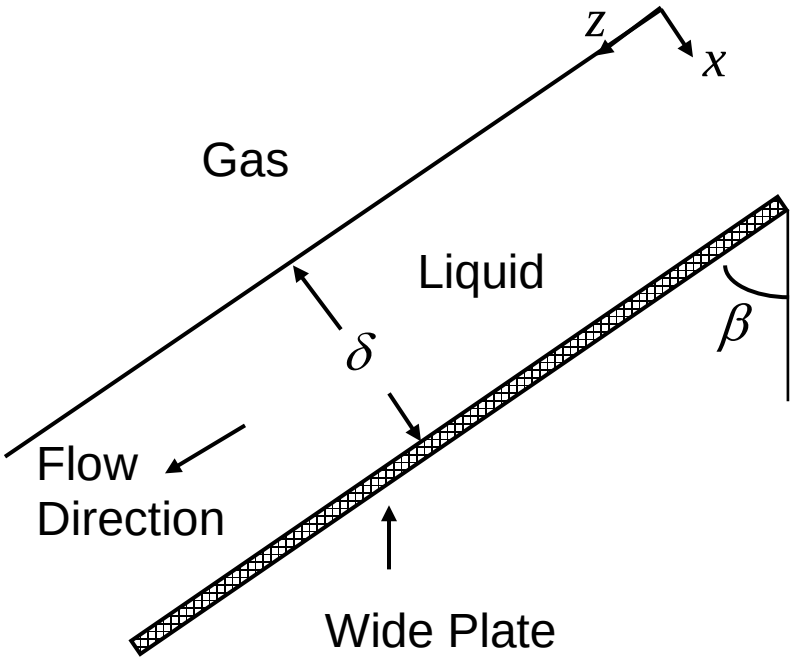
Part III

Example Problems in Fluid Flow

Flow in a Falling Liquid Film
Flow Through a Circular Tube
Flow Between Rotating Cylinders

Example 1. Flow in a Falling Liquid Film (from Bird et al. Section 2.2)

This first example is adapted from a worked-out problem in the textbook by Bird, Stewart, and Lightfoot. We consider laminar flow in a liquid film that flows down the inclined surface of a wide plate. Such flows are encountered in applications such as mass transfer from a gas to a liquid, and evaporation and condensation heat transfer. Our objective is to construct a simple model of this flow situation by making assumptions. Even though liquid films can have waves appearing on their surface, this is too much of a complication to accommodate at this stage. So, we shall assume that the flow is sufficiently slow that the free surface of the film is planar. For this geometry, rectangular Cartesian coordinates are an appropriate choice for convenience in satisfying the boundary conditions. A sketch of the system is given below.



Bird et al. use a shell balance approach to obtain the equation governing the velocity field. Here, we shall make a set of simplifying assumptions that will permit us to obtain the same governing equation from the full Navier-Stokes equations in component form. The advantage of our approach is that when we are finished, we'll know the specific assumptions that permit us to discard terms in the Navier-Stokes equations. Then, if we decide to relax one of these assumptions later, we can include the terms that were neglected based on that assumption. Also, the shell balance approach is limited in scope, as explained by Bird et al. in Section 2.1.

Assumptions

1. Steady laminar flow: $\frac{\partial \boldsymbol{v}}{\partial t} \equiv \boldsymbol{0}$

2. Incompressible flow (implies constant density ρ): the continuity equation reduces to $\nabla \bullet \boldsymbol{v} = 0$.

3. Newtonian flow at constant viscosity μ: this implies isothermal conditions and constant composition. In a problem involving heat or mass transport, the variation of viscosity because of any variations in temperature or composition should be relatively small for this to be a good assumption.

4. Negligible edge effects: this implies a system that is wide in the $y-$direction, normal to the plane of the paper in the sketch on the previous page. By neglecting edge effects, we are able to set $v_y \equiv 0$; $\frac{\partial \boldsymbol{v}}{\partial y} \equiv \boldsymbol{0}$.

5. Neglect end effects: this permits us to assume that the flow is fully developed. In the present situation, this means that the velocity distribution does not change with axial location along the film. Therefore, $\frac{\partial \boldsymbol{v}}{\partial z} \equiv \boldsymbol{0}$.

Note the use of the "identically equal to" sign in several places in this first example. This is done to highlight the fact that we can differentiate the left side with respect to a variable such as a position coordinate, and expect the derivative also to be zero everywhere. After we become comfortable with these assumptions, we can start using a simple equality sign.

Now, we begin by simplifying the continuity equation. In rectangular Cartesian coordinates, it is

$$\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0$$

(4) (5)

Note that we have crossed out the second and third terms. The reason for each is a specific assumption, the number assigned to which is shown above the term that is being crossed out. We see that continuity reduces to

$$\frac{\partial v_x}{\partial x} = 0$$

Now, from assumptions 1, 4, and 5, we already know that v_x cannot depend on t, y, and z, respectively, and therefore it is only a function of x. Now, we see that it can at best be just a constant. To establish the value of this constant, we invoke the kinematic boundary condition at

the bottom of the film, corresponding to $x = \delta$, namely, $v_x(\delta) = 0$. Because v_x must be a constant across the film, its value is zero everywhere.

$$\boxed{v_x \equiv 0}$$

Now, we proceed to simplify the (vector) Navier-Stokes equation written in component form in rectangular Cartesian coordinates, subject to assumptions 2 and 3. First, consider the $x-$ momentum equation.

$$\rho\left(\cancel{\frac{\partial v_x}{\partial t}} + \cancel{v_x}\frac{\partial v_x}{\partial x} + v_y\cancel{\frac{\partial v_x}{\partial y}} + v_z\cancel{\frac{\partial v_x}{\partial z}}\right) = -\frac{\partial p}{\partial x} + \rho g_x + \mu\left(\cancel{\frac{\partial^2 v_x}{\partial x^2}} + \cancel{\frac{\partial^2 v_x}{\partial y^2}} + \cancel{\frac{\partial^2 v_x}{\partial z^2}}\right)$$

[1] continuity [4] [5] continuity

Note that all the inertia terms are zero. The reason for setting each to zero is indicated above the term being crossed out. You'll find that sometimes it is possible to set a term to zero for more than one reason. On the right side, all the viscous terms are zero because, from continuity, $v_x \equiv 0$. In homework assignments and exams, it is important to always indicate the reason for crossing out any term as shown here.

The $x-$ momentum equation reduces to a simple hydrostatic balance.

$$\boxed{\frac{\partial p}{\partial x} = \rho g_x}$$

This should not be a surprise, because there is no velocity component in the $x-$ direction. Because $g_x = g\sin\beta$, we can rewrite the above result as

$$\boxed{\frac{\partial p}{\partial x} = \rho g \sin\beta}$$

Likewise, the $y-$ momentum equation also reduces to a hydrostatic balance.

$$\rho\left(\cancel{\frac{\partial v_y}{\partial t}} + v_x\cancel{\frac{\partial v_y}{\partial x}} + v_y\cancel{\frac{\partial v_y}{\partial y}} + v_z\cancel{\frac{\partial v_y}{\partial z}}\right) = -\frac{\partial p}{\partial y} + \rho g_y + \mu\left(\cancel{\frac{\partial^2 v_y}{\partial x^2}} + \cancel{\frac{\partial^2 v_y}{\partial y^2}} + \cancel{\frac{\partial^2 v_y}{\partial z^2}}\right)$$

[1] [4] [4] [4] [4] [4] [4]

$$\boxed{\frac{\partial p}{\partial y} = \rho g_y}$$

If the $y-$direction is normal to the gravity vector, $g_y = 0$, so that the pressure will be independent of the $y-$coordinate. Noting that pressure is also independent of time because of the assumption of steady state, we see that at best, $p = p(x,z)$ only. By integrating $\partial p / \partial x = \rho g \sin\beta$, we obtain

$$p(x,z) = p_0(z) + \rho g\, x \sin\beta$$

where p_0 can depend on z in general. By evaluating the pressure field at $x = 0$, we see that $p_0(z)$ is the pressure in the liquid at the free surface. Because the interface is flat, this is also the pressure in the gas phase at the free surface. There will be a slight hydrostatic variation in the gas pressure along the free surface because of the change in elevation, but given the very small density of a gas, we can ignore such variation in the present context. Therefore, the gas pressure at $x = 0$ is uniform in the $z-$direction. Thus, we conclude that p_0 is the (constant) atmospheric pressure above the liquid film, and is independent of z. This means that $\dfrac{\partial p}{\partial z}$ in the $z-$momentum equation can be set equal to zero.

Now, we proceed to the important component of the Navier-Stokes equation, namely the $z-$component.

1 continuity 4 5 shown above 4 5

$$\rho\left(\cancel{\frac{\partial v_z}{\partial t}} + \cancel{v_x}\frac{\partial v_z}{\partial x} + \cancel{v_y}\frac{\partial v_z}{\partial y} + v_z\cancel{\frac{\partial v_z}{\partial z}}\right) = -\cancel{\frac{\partial p}{\partial z}} + \rho g_z + \mu\left(\frac{\partial^2 v_z}{\partial x^2} + \cancel{\frac{\partial^2 v_z}{\partial y^2}} + \cancel{\frac{\partial^2 v_z}{\partial z^2}}\right)$$

The component of the gravity vector in the $z-$direction is $g\cos\beta$ so that the above equation can be rewritten as

$$\frac{d^2 v_z}{dx^2} = -\frac{\rho g \cos\beta}{\mu}$$

You'll note that we have replaced the partial derivative sign in the $z-$component momentum equation with the ordinary derivative sign because v_z can depend only upon x. The reason is that from assumptions 1,4, and 5, it is independent of t, y, and z, respectively. Also note that the entire right side of the above differential equation for v_z is a constant. Therefore, we can integrate this equation twice in a straightforward manner to yield

$$v_z(x) = -\frac{\rho g \cos\beta}{2\mu} x^2 + C_1 x + C_2$$

The two constants of integration can be evaluated using boundary conditions on the velocity field. The no-slip condition can be used at the interface with the solid, and at the free gas-liquid interface,

the velocity gradient (and therefore, the shear stress) is negligible. Therefore, the following boundary conditions can be written.

$v_z(\delta) = 0$ no slip

$\frac{dv_z}{dx}(0) = 0$ negligible shear stress

Note how we write the physical reason for each boundary condition immediately after it. This will serve to remind us in a compact manner why each boundary condition is valid.

Applying these boundary conditions leads to the following velocity distribution in the liquid film.

$$v_z(x) = \frac{\rho g \delta^2 \cos\beta}{2\mu}\left(1 - \frac{x^2}{\delta^2}\right)$$

The textbook provides a sketch of the velocity profile across the film, as well as a sketch of the shear stress.

The volumetric flow rate of liquid is a quantity of interest. The elementary volumetric flow rate dQ across an area element of depth dx and width W in the $y-$direction is the product of the velocity at that location $v_z(x)$ and the area of the element Wdx. Adding up all the elementary volumetric flow rates across the depth of the film gives the total volumetric flow rate. This summation process for differential elements is simply an integration over the depth of the film. The total volumetric flow rate Q can be written as

$$Q = \int_0^\delta v_z(x) W dx = \boxed{\frac{\rho g \delta^3 W \cos\beta}{3\mu}}$$

Note that the volumetric flow rate is proportional to the cube of the film thickness. Such relationships are called "scalings" in a given situation, and are valuable pieces of information about a physical system. Sometimes, it is possible to infer such scalings from dimensional arguments.

The average velocity is defined as the ratio of the volumetric flow rate to the cross-sectional area available for flow. Therefore,

$$v_{z,avg} = \frac{Q}{W\delta} = \boxed{\frac{\rho g \delta^2 \cos\beta}{3\mu}}$$

It is interesting to note that the maximum velocity in the film occurs at the free surface where the slope of the velocity field is zero. From calculus, we know that the maximum or minimum of a function occurs at a location where the slope is zero. But we need to be careful here, because the

maximum actually occurs at the end point of the interval in x. It is not necessary for the slope to be equal to zero for the maximum (or minimum) of a function to be reached at an end point of the interval in which the function is defined.

By evaluating the result for the velocity distribution at $x = 0$, we find the maximum velocity to be

$$v_{z,max} = \boxed{\frac{\rho g \delta^2 \cos\beta}{2\mu}}$$

so that the average velocity in the film is 2/3 the maximum value. This result also holds true for steady, fully developed, incompressible Newtonian laminar flow between two parallel plates. The reason is that the velocity profile in that problem between one of the plates and the central plane is precisely the same as the one we have established for the film. This is because the governing equation for the velocity distribution is identical to that we obtained here, and the boundary conditions at one of the solid surfaces and the central plane are also identical to those applicable to the film flow problem.

Here are some questions to ponder.

1. How would the model be affected in the presence of standing waves on the surface? How about moving waves?

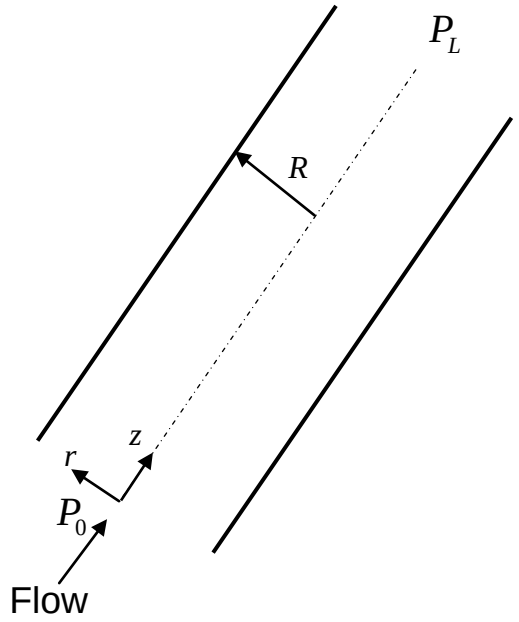
2. How far from the inlet and the exit do we have to be for this analysis of fully developed flow to be valid? How does this distance depend on the properties of the fluid and the average velocity?

3. In a like manner, how far from the edges do we need to go for the assumption of negligible edge effects to be valid?

4. In photographic film manufacture, multiple liquid layers are commonly encountered. Try posing the problem for two liquid layers flowing down an inclined plate.

Example 2. Flow Through a Circular Tube (from Bird et al. Section 2.3)

This example is adapted from a worked-out problem in the textbook by Bird, Stewart, and Lightfoot. The objective is to obtain a description of the details of the laminar flow that occurs in a straight circular tube under the action of a dynamic pressure difference between the inlet and the exit. Circular tubes in which flow occurs are found a variety of applications, starting from household plumbing to industrial pipes. Also, the blood in our bodies flows through vessels that can be roughly approximated as circular tubes. In the present problem, we make several simplifying assumptions that make the problem tractable at this stage. Nevertheless, the results are of some practical use. A sketch of the system is given below.



The dynamic pressure is P_0 at the inlet $z = 0$, and P_L at the exit $z = L$. As in Example 1, Bird et al. use a shell balance approach to obtain the equation governing the velocity field. Here, we shall make a set of simplifying assumptions that will permit us to obtain the same governing equation from the full Navier-Stokes equations in component form.

Assumptions

1. Steady laminar flow: $\frac{\partial \boldsymbol{v}}{\partial t} \equiv \boldsymbol{0}$

2. Incompressible flow (implies constant density ρ): the continuity equation reduces to $\nabla \bullet \boldsymbol{v} = 0$.

3. Newtonian flow at constant viscosity μ: this implies isothermal conditions and constant composition.

4. Symmetry about the axis. $v_\theta \equiv 0;\ \ \dfrac{\partial \boldsymbol{v}}{\partial \theta} \equiv 0$

5. System is long in the $z-$direction – therefore, neglect end effects. This permits us to assume that the flow is fully developed. This means that the velocity distribution does not change with axial position along the tube. Therefore, $\dfrac{\partial \boldsymbol{v}}{\partial z} \equiv \boldsymbol{0}$.

First, we simplify the continuity equation.

$$\frac{1}{r}\frac{\partial}{\partial r}\left(rv_r\right) + \frac{1}{r}\frac{\partial \cancel{v_\theta}}{\partial \theta} + \frac{\partial \cancel{v_z}}{\partial z} = 0 \qquad \boxed{4} \quad \boxed{5}$$

Therefore, we conclude that the product rv_r must be independent of the radial coordinate r. Because of assumptions 1, 4, and 5, v_r is independent of time and the spatial coordinates θ and z, and therefore rv_r must be a constant. From the kinematic condition at the tube wall, $v_r\left(R\right) = 0$. Hence the function rv_r is zero at the wall, and must assume this value everywhere. Therefore, we conclude that $v_r \equiv 0$ everywhere. We already know that $v_\theta \equiv 0$ from assumption 4. Therefore, this is a unidirectional flow with the velocity given by $\boldsymbol{v} = \boldsymbol{k}v_z$, where $\boldsymbol{k}$ is a unit vector in the $z-$direction.

Now, we proceed to simplify the components of the Navier-Stokes equation in cylindrical polar coordinates.

$r-$component of the Navier-Stokes equation

$$\rho\left[\frac{\partial \cancel{v_r}}{\partial t} + v_r\frac{\partial \cancel{v_r}}{\partial r} + \frac{v_\theta}{r}\frac{\partial \cancel{v_r}}{\partial \theta} - \frac{\cancel{v_\theta^2}}{r} + v_z\frac{\partial \cancel{v_r}}{\partial z}\right] =$$

$$\boxed{1} \quad \boxed{\text{continuity}} \qquad \boxed{4} \quad \boxed{\text{continuity}}$$

$$-\frac{\partial P}{\partial r} + \mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\cancel{v_r}\right)\right) + \frac{1}{r^2}\frac{\partial^2 \cancel{v_r}}{\partial \theta^2} - \frac{2}{r^2}\frac{\partial \cancel{v_\theta}}{\partial \theta} + \frac{\partial^2 \cancel{v_r}}{\partial z^2}\right]$$

$$\boxed{\text{continuity}} \qquad \boxed{4} \quad \boxed{\text{continuity}}$$

Therefore, we conclude that $\dfrac{\partial P}{\partial r} = 0$, where P is the dynamic pressure.

$\theta-$component of the Navier-Stokes equation

$$\rho\left[\frac{\partial v_\theta}{\partial t}+v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}+\frac{v_r v_\theta}{r}+v_z\frac{\partial v_\theta}{\partial z}\right]=$$

$$-\frac{1}{r}\frac{\partial P}{\partial \theta}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r v_\theta\right)\right)+\frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2}+\frac{2}{r^2}\frac{\partial v_r}{\partial \theta}+\frac{\partial^2 v_\theta}{\partial z^2}\right]$$

Hence, $\frac{\partial P}{\partial \theta}=0$.

$z-$component of the Navier-Stokes equation

$$\rho\left[\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_z}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta}+v_z\frac{\partial v_z}{\partial z}\right]=$$

$$-\frac{\partial P}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2}+\frac{\partial^2 v_z}{\partial z^2}\right]$$

Recognizing from assumptions 1, 4, and 5 that $v_z=v_z(r)$ only, and from assumption 1 and the r and $\theta-$components of the Navier-Stokes equation that $P=P(z)$ only, we are able to write the following equation, where the partial derivatives have been replaced by ordinary derivatives.

$$\mu\frac{1}{r}\frac{d}{dr}\left(r\frac{dv_z}{dr}\right)=\frac{dP}{dz}$$

Note that the left side can depend only on r, whereas the right side can depend only on z. This is possible only if each side is a constant that is independent of either r or z. Therefore,

$$\mu\frac{1}{r}\frac{d}{dr}\left(r\frac{dv_z}{dr}\right)=\frac{dP}{dz}=A$$

where A is a constant. This means that the dynamic pressure must vary linearly with z in the tube, with a constant slope A. We can determine this slope as

$$A=\frac{P_L-P_0}{L}=-\frac{\Delta P}{L}$$

where $\Delta P = P_0 - P_L$. Hence the differential equation satisfied by the velocity distribution can be written as

$$\frac{1}{r}\frac{d}{dr}\left(r\frac{dv_z}{dr}\right) = -\frac{\Delta P}{\mu L}$$

One integration is performed after rewriting the equation as

$$\frac{d}{dr}\left(r\frac{dv_z}{dr}\right) = -\frac{\Delta P}{\mu L}\ r$$

to yield

$$r\frac{dv_z}{dr} = -\frac{\Delta P}{2\mu L}\ r^2 + c_1$$

where c_1 is a constant of integration. Rearrange this equation as $\frac{dv_z}{dr} = -\frac{\Delta P}{2\mu L}\ r + \frac{c_1}{r}$ and integrate once more to obtain

$$v_z(r) = -\frac{\Delta P}{4\mu L}\ r^2 + c_1 \ln r + c_2$$

where we have introduced a second constant of integration c_2. We'll need two boundary conditions to evaluate these constants. The boundary conditions are

$v_z(0)$ is finite

$v_z(R) = 0$ No slip at the wall

The condition at the centerline is sufficient to eliminate one of the constants of integration. An alternative boundary condition that is used at the centerline is

$$\frac{dv_z}{dr}(0) = 0$$

This is called a "symmetry condition." This condition can be derived by examining the force balance on a cylinder of fluid of radius ε sharing the same axis as the tube. The sum of the forces acting on this cylinder must be zero, because there is no acceleration. The two forces acting on it are the pressure force $\Delta P \times \pi\varepsilon^2$ and the shear exerted by the adjacent fluid over the surface of the cylinder, which is given by $\tau_{rz}(\varepsilon) \times 2\pi\varepsilon L$. Setting the sum of these forces to zero, and substituting $\tau_{rz} = \mu\frac{dv_z}{dr}$, we obtain the following result.

$$\frac{dv_z}{dr}(\varepsilon) = -\frac{\Delta P}{2\mu L}\varepsilon$$

By taking the limit as $\varepsilon \to 0$, we obtain the alternative boundary condition given earlier.

Integrating the governing differential equation for $v_z(r)$ and applying the two boundary conditions leads to the following velocity profile. A sketch can be found in Fig. 2.3-2 in the text.

$$\boxed{v_z(r) = \frac{\Delta P R^2}{4\mu L}\left(1 - \frac{r^2}{R^2}\right)}$$

Maximum velocity

Because of the symmetry in the system, which requires the velocity to increase from a value of zero at the tube wall toward the centerline in the same manner for all values of θ, it is evident that the maximum velocity occurs at the centerline, $r = 0$. This can also be observed by examining the dependence of the velocity distribution on the radial coordinate.

The result for the maximum velocity is

$$\boxed{v_{z,\max} = \frac{\Delta P R^2}{4\mu L}}$$

Note that the argument that the derivative $\frac{dv_z}{dr}$ can be set equal to zero to find the location of the maximum velocity cannot be used here. This is because the maximum occurs at a boundary of the domain, $r = 0$, and the slope at the boundary need not be zero for an extremum in a function to occur at the boundary.

Volumetric and Mass Flow Rates

The volumetric flow rate is obtained by using $Q = \int_A v_z \, dA$ where A is the cross-sectional area.

$$\boxed{Q = \int_0^R v_z(r)\, 2\pi r dr = \frac{\pi \Delta P R^4}{8\mu L}}$$

The mass flow rate is given as $w = \rho Q$.

Average Velocity

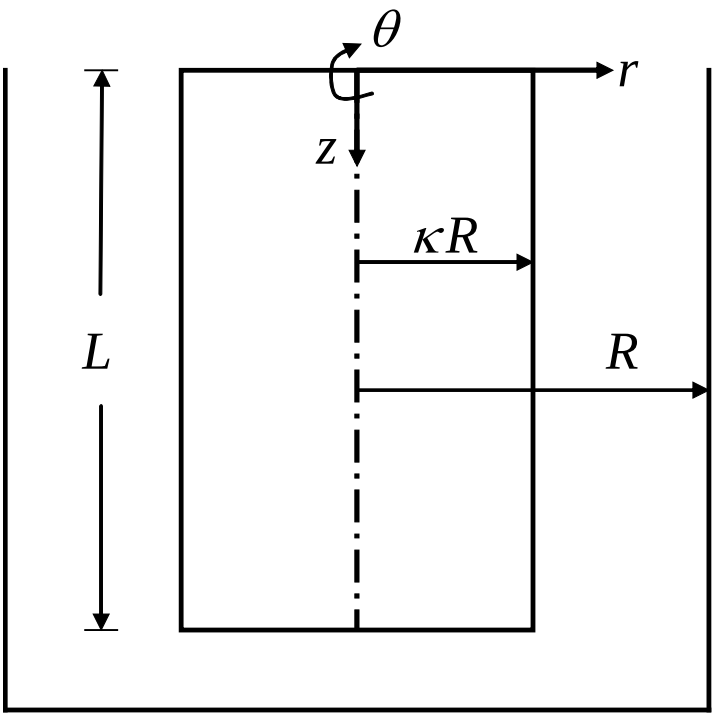
The average velocity is the ratio of the volumetric flow rate to the cross-sectional area.

$$v_{z,avg} = \frac{Q}{\pi R^2} = \frac{\pi \Delta P R^4 / (8\mu L)}{\pi R^2} = \boxed{\frac{\Delta P R^2}{8\mu L}}$$

Note that in the case of the circular tube, the average velocity is one-half the maximum, whereas in the case of flow between parallel plates (and in a film flowing down a solid surface) the average is two-third of the maximum. These are simply the quirks of the geometries involved.

Example 3. Flow Between Rotating Cylinders (from Bird et al. Example 3.6-3)

This example is similar to that worked out in the textbook by Bird, Stewart, and Lightfoot. We consider laminar flow in a fluid that occupies the annular space between two concentric vertical cylinders. The outer cylinder is rotated at a constant angular speed Ω_o while the inner cylinder is rotated at a constant angular speed Ω_i. Such flows are encountered in viscometers, which are used for measuring the viscosity of fluids. As in Example 1, our objective is to construct a simple model of this flow situation by making assumptions. For this geometry, cylindrical polar coordinates are an appropriate choice for convenience in satisfying the boundary conditions. A sketch of a side view of the system is given below. You can also consult the textbook for additional sketches, such as a plan view.



Next, we list a set of simplifying assumptions.

Assumptions

1. Steady laminar flow: $\frac{\partial \boldsymbol{v}}{\partial t} \equiv \boldsymbol{0}$

2. Incompressible flow (implies constant density ρ): the continuity equation reduces to $\nabla \bullet \boldsymbol{v} = 0$

.

3. Newtonian flow at constant viscosity μ: this implies isothermal conditions and constant composition. If the viscosity of the fluid is large and the shear rate is large, viscous dissipation can lead to heating of the fluid. The viscometer will need to be kept in a thermostat in such a situation to maintain isothermal conditions.

4. The flow is symmetrical about the axis. This, of course, requires that the inner cylinder be perfectly centered within the outer cylinder. With this assumption, we are able to set $\frac{\partial \mathbf{v}}{\partial \theta} \equiv \mathbf{0}$. Also, the dynamic pressure will be independent of the angular coordinate θ.

5. Neglect end effects: this means that there should be no $z-$component of the velocity, and furthermore that the velocity distribution does not change with axial location along the annulus. Therefore, $v_z \equiv 0$; $\frac{\partial \mathbf{v}}{\partial z} \equiv \mathbf{0}$.

Now, we begin by simplifying the continuity equation. In cylindrical polar coordinates, it is

$$\frac{1}{r}\frac{\partial}{\partial r}\left(rv_r\right) + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} + \frac{\partial v_z}{\partial z} = 0 \qquad \boxed{4} \quad \boxed{5}$$

Note that we have crossed out the second and third terms. The specific assumption used for crossing out a given term is listed above that term. Continuity reduces to

$$\frac{1}{r}\frac{\partial}{\partial r}\left(rv_r\right) = 0$$

From assumptions 1, 4, and 5, we already know that v_r cannot depend on t, θ, and z, respectively, and therefore it is only a function of r. We can integrate the above simplified version of the continuity equation to obtain the following result.

$$rv_r = C$$

Here, C is just a constant. To establish the value of this constant, we invoke the kinematic boundary condition at either of the two boundaries in the radial coordinate. Choosing $r = R$ for this purpose, we can write $v_r(R) = 0$. Therefore, the constant $C = 0$. You can see, of course, that the same result would have been obtained if we had invoked the kinematic boundary condition at the other boundary $r = \kappa R$. As a consequence, we find that the radial component of the velocity field is identically zero.

$$\boxed{v_r \equiv 0}$$

Note that from assumption 5, the axial component $v_z \equiv 0$. Therefore, the only non-zero component is v_θ. From assumptions 1, 4, and 5, we can see that v_θ is independent of time t and the position coordinates θ and z, respectively. Therefore, $v_\theta = v_\theta(r)$ only, and our quest now is to establish the nature of this dependence. For this, we need to go to the Navier-Stokes equations in component form in cylindrical polar coordinates. Given assumptions 2 and 3, we can look up these component equations from Table B.6 of Bird et al. First, we briefly consider the components in the r and z directions. In the momentum equations, we use $\mathcal{P}$ to designate the dynamic pressure. The $r-$ momentum equation is simplified as follows.

$$\underset{\boxed{1}\ \text{continuity}}{\rho\left[\frac{\partial v_r}{\partial t} + v_r\frac{\partial v_r}{\partial r} + \frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta} - \frac{v_\theta^2}{r} + v_z\frac{\partial v_r}{\partial z}\right]} \underset{\boxed{4}\quad\boxed{5}}{=}$$

$$\underset{\text{continuity}\quad\text{continuity}\quad\boxed{4}\quad\text{continuity}}{-\frac{\partial \mathcal{P}}{\partial r} + \mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(rv_r)\right) + \frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2} - \frac{2}{r^2}\frac{\partial v_\theta}{\partial \theta} + \frac{\partial^2 v_r}{\partial z^2}\right]}$$

The $r-$momentum equation reduces to a gyrostatic balance.

$$\boxed{\frac{\partial \mathcal{P}}{\partial r} = \rho\frac{v_\theta^2}{r}}$$

You might wonder why there is a pressure gradient in the radial direction. Recall that Newton's law tells us that a body will move at a constant velocity in a straight line unless acted upon by an external force. Fluid elements in the annular space are not moving in straight lines, but rather along circular paths. Some force has to act on a fluid element that pushes it radially inward so that it will move along a circle. This force arises from this pressure gradient, which develops naturally in this geometry. The dynamic pressure increases radially outward at precisely the rate needed to keep the fluid going around along circular streamlines.

Now, consider the $z-$momentum equation.

$$\underset{\boxed{1}\quad\boxed{5}\quad\boxed{5}\quad\boxed{5}}{\rho\left[\frac{\partial v_z}{\partial t} + v_r\frac{\partial v_z}{\partial r} + \frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta} + v_z\frac{\partial v_z}{\partial z}\right]} =$$

$$\underset{\boxed{5}\quad\boxed{5}\quad\boxed{5}}{-\frac{\partial \mathcal{P}}{\partial z} + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2} + \frac{\partial^2 v_z}{\partial z^2}\right]}$$

It simplifies to

$$\boxed{\frac{\partial \mathcal{P}}{\partial z} = 0}$$

Therefore, the dynamic pressure is uniform in the $z-$direction, or the only variation of the actual pressure in that direction is hydrostatic.

Finally, we proceed to the $\theta-$component of the Navier-Stokes equation.

$$\begin{array}{c} \boxed{1}\ \text{continuity} \qquad \boxed{4}\ \text{continuity}\ \boxed{5} \\ \rho\left[\cancel{\frac{\partial v_\theta}{\partial t}} + \cancel{v_r}\frac{\partial v_\theta}{\partial r} + \frac{v_\theta}{r}\cancel{\frac{\partial v_\theta}{\partial \theta}} + \frac{\cancel{v_r} v_\theta}{r} + \cancel{v_z}\frac{\partial v_\theta}{\partial z}\right] = \\ \boxed{4} \qquad\qquad\qquad\qquad \boxed{4}\ \text{continuity}\ \boxed{5} \\ -\frac{1}{r}\cancel{\frac{\partial \mathcal{P}}{\partial \theta}} + \mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r v_\theta\right)\right) + \frac{1}{r^2}\cancel{\frac{\partial^2 v_\theta}{\partial \theta^2}} + \frac{2}{r^2}\frac{\partial \cancel{v_r}}{\partial \theta} + \cancel{\frac{\partial^2 v_\theta}{\partial z^2}}\right] \end{array}$$

Therefore, the $\theta-$component simplifies to

$$\frac{d}{dr}\left(\frac{1}{r}\frac{d}{dr}\left(r v_\theta\right)\right) = 0$$

You'll note that we have replaced the partial derivative sign in the $\theta-$component momentum equation with the ordinary derivative sign. The first integral of this equation is

$$\frac{1}{r}\frac{d}{dr}\left(r v_\theta\right) = C_1$$

A second integration yields

$$v_\theta = \frac{C_1}{2} r + \frac{C_2}{r}$$

The two boundary conditions on the velocity field are

$$v_\theta\left(\kappa R\right) = \Omega_i \kappa R \qquad \text{no slip}$$
$$v_\theta\left(R\right) = \Omega_o R \qquad \text{no slip}$$

The two constants of integration can be evaluated using these boundary conditions. The values are

$$C_1 = 2\frac{\Omega_o - \kappa^2\Omega_i}{1-\kappa^2} \qquad C_2 = \frac{\Omega_i - \Omega_o}{1-\kappa^2}\kappa^2 R^2$$

Substituting these results into the velocity distribution and rearranging leads to the following solution for the velocity field.

$$v_\theta(r) = \frac{R}{1-\kappa^2}\left[\left(\Omega_o - \kappa^2\Omega_i\right)\frac{r}{R} + \kappa^2\left(\Omega_i - \Omega_o\right)\frac{R}{r}\right]$$

Try sketching the velocity distribution qualitatively correctly. This means that the end values should be correct and that the slope should vary across the gap in the correct manner.

In a viscometer, typically one cylinder is held fixed while the other is rotated. The torque required to hold the chosen cylinder fixed is measured, and using the theoretical result, the viscosity can be evaluated. Let us assume the inner cylinder is held fixed and the torque acting on it is measured. In this case, we can set $\Omega_i = 0$ in the velocity distribution to obtain

$$v_\theta(r) = \frac{\Omega_o R}{1-\kappa^2}\left[\frac{r}{R} - \frac{\kappa^2 R}{r}\right]$$

In the present case, the measured torque is that exerted by the shear force acting on the surface of the inner cylinder about the axis. If the magnitude of the force is F, then the magnitude of the torque is given by $T = \kappa RF$.

To find the shear force on the inner cylinder, we first evaluate the shear stress. A general result for the shear stress $\tau_{r\theta} = \tau_{\theta r}$ can be found from Table B.1 in the textbook.

$$\tau_{r\theta} = \tau_{\theta r} = \mu\left[r\frac{\partial}{\partial r}\left(\frac{v_\theta}{r}\right) + \frac{1}{r}\frac{\partial v_r}{\partial\theta}\right]$$

Note that our sign convention for stress is the opposite of that used by Bird et al. This is the reason for removing the negative sign in the result in Table B.1. Because $v_r \equiv 0$, the above result for the shear stress simplifies to

$$\tau_{r\theta} = \mu\left[r\frac{\partial}{\partial r}\left(\frac{v_\theta}{r}\right)\right] = 2\mu\,\Omega_o\frac{\kappa^2}{1-\kappa^2}\frac{R^2}{r^2}$$

Even though we only need the shear stress at the surface of the inner cylinder for evaluating the desired torque, let us, for the present, make a general evaluation of the torque on a cylinder of radius r anywhere in the annular gap.

The area on which the shear stress acts is $2\pi rL$, and the lever arm is r, so that the torque is given by

$$T = 2\pi r^2 L\tau_{r\theta} = 4\pi\mu\,\Omega_o R^2 L\frac{\kappa^2}{1-\kappa^2}$$

We see that this torque is independent of the radius of the cylinder to which it is applied, and is uniform across the gap.

Bird et al. give a good discussion of some issues associated with this flow as the Reynolds number is gradually increased. For example, the assumption that $v_z \equiv 0$, while plausible, is not consistent with observation when the angular velocity exceeds a critical value for a given fluid and geometrical parameters. A secondary flow appears that causes the flow to break into cells, known as Taylor vortices. This fact points to an important consequence of the non-linearity of the Navier-Stokes equation. Non-linear equations can have multiple solutions, and which of these is realized physically depends on the stability of each solution to small perturbations that are always present. To learn more about this problem, you can consult the book by Koschmieder (1993).

References

1. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, Transport Phenomena, Wiley, 2007.

2. E.L. Koschmieder, Bénard Cells and Taylor Vortices, Cambridge University Press, 1993.

Part IV

Mathematical Techniques: Solution of Partial Differential Equations

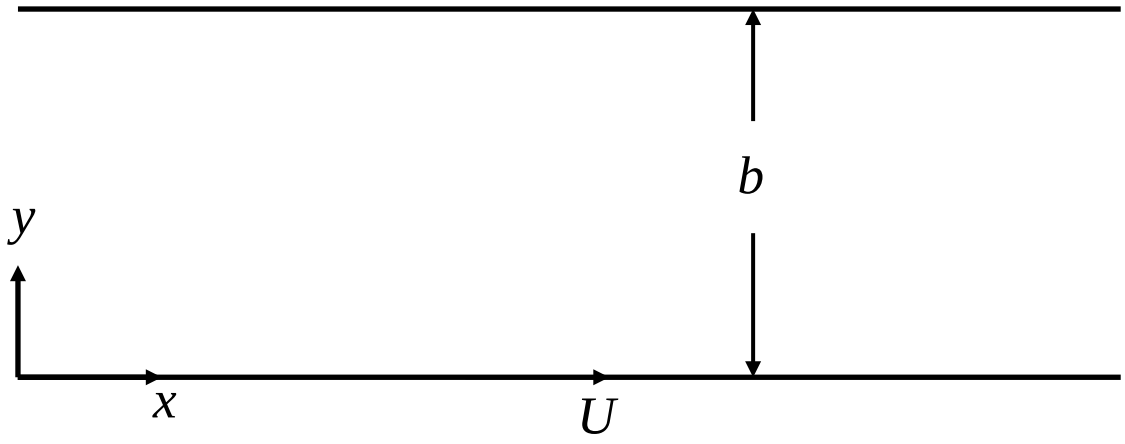
Separation of Variables
Combination of Variables

Solution of Partial Differential Equations

Separation of Variables

Introduction and Problem Statement

We encounter partial differential equations routinely in transport phenomena. Some examples are unsteady flow in a channel, steady heat transfer to a fluid flowing through a pipe, and mass transport to a falling liquid film. Here, we shall learn a powerful method for solving many of these partial differential equations. We shall also learn when the method can be used. The model problem we consider is the motion induced in fluid contained between two long and wide parallel plates placed with a distance b between them as shown in the sketch below.



The fluid is initially assumed to be at rest. Motion is initiated by suddenly moving the bottom plate at a constant velocity of magnitude U in the $x-$direction. The velocity of the bottom plate is maintained at that value for all future values of time t while the top plate is held fixed in place. There is no applied pressure gradient, with motion being caused strictly by the movement of the bottom plate.

We shall assume the flow to be incompressible with a constant density ρ and Newtonian with a constant viscosity μ. We neglect edge effects in the $z-$direction so that we can set $v_z = 0$ and $\frac{\partial \boldsymbol{v}}{\partial z} = 0$, and assume fully developed flow, implying $\frac{\partial \boldsymbol{v}}{\partial x} = 0$. Here, $\boldsymbol{v}$ stands for the velocity vector, and the subscripts denote components.

It can be established from the continuity equation and the kinematic condition at one of the walls that $v_y = 0$. Therefore, the only non-zero velocity component is $v_x(t, y)$, which can be shown to satisfy the following partial differential equation.

$$\frac{\partial v_x}{\partial t} = \nu \frac{\partial^2 v_x}{\partial y^2} \tag{1}$$

In Equation (1), t represents time, and ν is the kinematic viscosity. The initial condition is

$$v_x(0, y) = 0 \tag{2}$$

and the boundary conditions are

$$v_x(t, 0) = U \tag{3}$$

and

$$v_x(t, b) = 0 \tag{4}$$

It is convenient to work with scaled variables. Scaling minimizes the number of parameters in the problem, and helps us identify the true (dimensionless) parameters so that we can perform asymptotic analyses where desired. Whenever possible, we use a reference quantity, termed a "scale" for the variable involved, that will normalize that variable, meaning that the range of values assumed by the dimensionless variable will be from 0 to 1. The natural variables that normalize the velocity and the y – coordinate in this problem are U and b, respectively. Therefore, we define a scaled velocity $V = \frac{v_x}{U}$ and a scaled distance variable $Y = \frac{y}{b}$. Introducing these definitions into the differential equation, we obtain

$$\frac{\partial V}{\partial t} = \frac{\nu}{b^2} \frac{\partial^2 V}{\partial Y^2} \tag{5}$$

which suggests that we might choose the time scale as b^2 / ν, defining a scaled time $T = \frac{\nu t}{b^2}$. Thus, Equation (5) can be rewritten as

$$\frac{\partial V}{\partial T} = \frac{\partial^2 V}{\partial Y^2} \tag{6}$$

and the initial and boundary conditions become

$$V(0, Y) = 0 \tag{7}$$

$$V(T, 0) = 1 \tag{8}$$

$$V(T, 1) = 0 \tag{9}$$

Now, we are ready to learn the mathematical technique of "Separation of Variables." The usual way to solve a partial differential equation is to find a technique to convert it to a system of ordinary differential equations. Then, we can use methods available for solving ordinary differential equations. One important requirement for separation of variables to work is that the governing partial differential equation and initial and boundary conditions be linear. Another is that for the class of partial differential equation represented by Equation (6), the boundary conditions in the $Y-$coordinate be homogeneous. This means that any constant times the dependent variable should satisfy the same boundary condition. Also, the differential equation itself should be homogeneous. A condition in which the variable or a linear combination of the variable and its spatial or time derivative is set equal to 0 can be seen to be a homogeneous condition.

We see that Equation (6) is homogeneous because a constant times V will satisfy the same equation. Equation (9) is homogeneous as well, but Equation (8) is not. Therefore, we must first define a new problem in which homogeneous boundary conditions can be written. The approach we follow is based on the physical aspects of the problem. Consider the same fluid mechanical problem at steady state, wherein we set the time derivative of the velocity equal to zero. This means that we can no longer expect to satisfy the initial condition, but the boundary conditions still hold. The resulting steady velocity field $V_s(Y)$ can be seen from Equations (6), (8), and (9) to satisfy

$$\frac{d^2V_s}{dY^2}=0 \tag{10}$$

$$V_s(0)=1 \tag{11}$$

$$V_s(1)=0 \tag{12}$$

The solution is seen to be

$$V_s(Y)=1-Y \tag{13}$$

Now, write the solution of Equations (6) - (9) as the sum of the above steady solution and a transient contribution that we expect will decay to zero as $T\to\infty$.

$$V(T,Y)=V_s(Y)+V_t(T,Y) \tag{14}$$

Equation (14) is substituted into Equations (6) - (9), and use is made of Equations (10) -(12). This yields the governing equation and the initial and boundary conditions for the transient field $V_t(T,Y)$.

$$\frac{\partial V_t}{\partial T}=\frac{\partial^2V_t}{\partial Y^2} \tag{15}$$

$$V_t(0,Y)=-V_s(Y) \tag{16}$$

$$V_t(T,0)=0 \tag{17}$$

$$V_t(T,1)=0 \tag{18}$$

It is seen that the inhomogeneity in the boundary condition for $V(T,0)$ has been taken up by $V_s(0)$, leaving us with a homogeneous boundary condition for $V_t(T,0)$. If the governing differential equation had a time-independent inhomogeneity, we can expect the same will happen. That inhomogeneity will be included in the governing equation for the steady field, leaving the governing equation for the transient field homogeneous.

Product Class Solution

Now, we attempt a solution of Equation (15) in the form of a product

$$V_t(T,Y) = G(T)\phi(Y) \tag{19}$$

This is not to suggest that the final solution will be exactly like this. It is a trial solution, just like the trial solution e^{mx} that is used in the case of a linear ordinary differential equation with constant coefficients. The approach will be to substitute this trial solution in the governing equation and the initial and boundary conditions to see if it might possibly satisfy them. First inserting it into Equation (15) yields

$$G'\phi = G\phi'' \tag{20}$$

where we have used primes to denote differentiation with respect to the argument of the function. Thus, G' stands for dG/dT whereas ϕ'' connotes $d^2\phi/dY^2$. Divide both sides of Equation (20) by $G\phi$. This yields

$$\frac{G'}{G} = \frac{\phi''}{\phi} \tag{21}$$

But, the left side of the above equation can depend only on T, whereas the right side can depend only on Y. How can it be possible for a function of only T to be equal to a function of only Y? The answer is: Never, unless we force both functions to be a constant that is independent of T and Y. For reasons that will become clear later, we require this "constant of separation" to be negative. So, we set it equal to $-\lambda^2$ where λ is a real number.

$$\frac{G'}{G} = \frac{\phi''}{\phi} = -\lambda^2 \tag{22}$$

So we see that we have made a lot of progress. We now have two ordinary differential equations in place of the partial differential equation. They are

$$G' + \lambda^2 G = 0 \tag{23}$$

and

$$\phi'' + \lambda^2 \phi = 0 \tag{24}$$

The solution of Equation (23) can be written as

$$G(T) = \alpha \exp\left[-\lambda^2 T\right] \tag{25}$$

where α is a constant of integration. Notice that Equation (25) implies that as $T \to \infty$, $G \to 0$, which is consistent with our idea that the transient solution will decay as $T \to \infty$. The general solution of Equation (24) for $\phi(Y)$ can be written as

$$\phi(Y) = c_1 \sin \lambda Y + c_2 \cos \lambda Y \tag{26}$$

where c_1 and c_2 are constants of integration. Because the boundary conditions on V_t at $Y = 0$ and $Y = 1$ that are given in Equations (17) and (18), respectively, are both homogeneous, they can be satisfied by setting

$$\phi(0) = 0 \tag{27}$$

$$\phi(1) = 0 \tag{28}$$

Application of these boundary conditions yields the following results.

$$c_2 = 0 \tag{29}$$

$$c_1 \sin \lambda = 0 \tag{30}$$

If we try to satisfy Equation (30) with the choice $c_1 = 0$, we obtain the result that $\phi(Y) \equiv 0$. This yields the trivial solution $V_t = 0$. This is incorrect because it does not satisfy the initial condition on V_t given in Equation (16). Therefore, we must choose the alternative

$$\sin \lambda = 0 \tag{31}$$

This equation has an infinite number of roots that occur in pairs.

$$\lambda = \lambda_n = \pm n\pi, \qquad n = 0, 1, 2, \ldots \tag{32}$$

First, we note that the case $n = 0$ can be discarded because it again leads to the trivial solution that is unacceptable. Second, the negative roots do not yield an independent solution because $\sin(-n\pi Y) = -\sin(n\pi Y)$. Therefore, we can write the solution for $\phi(Y)$ as

$$\phi(Y) = \phi_n(Y) = c_n \sin \lambda_n Y \tag{33}$$

with

$$\lambda_n = n\pi, \qquad n = 1, 2, 3, \ldots \tag{34}$$

Note that we have replaced the single constant c_1 with a subscripted constant c_n to underscore the fact that each of these acceptable solutions can be multiplied by a different arbitrary constant.

The net result of the exercise has been to produce an infinite set of product class solutions for V_t. By representing the product of the arbitrary constants α and c_n using a new constant A_n, we can write the n'th solution as $A_n e^{-\lambda_n^2 T} \sin \lambda_n Y$. At $T = 0$, this becomes $A_n \sin \lambda_n Y$. This is a periodic function and does not at all look like the function $-V_s(Y)$, which happens to be a straight line in the interval $[0,1]$. Fortunately, because the governing equation and boundary conditions are linear and homogeneous, we can add all of these solutions and try to see if the sum can be used to satisfy the initial condition by judicious choice of the constants A_n. Therefore, we write

$$V_t(T,Y) = \sum_{n=1}^{\infty} A_n \; e^{-\lambda_n^2 T} \sin \lambda_n Y \tag{35}$$

Notice that there is no problem when we add a finite number of solutions, but when the upper limit of summation is infinity, we need to concerned with the issue of whether the right side converges. Such mathematical issues are considered in detail in Weinberger [1]. Here, we assume that the sum uniformly converges for all values of scaled time T and all values of Y in the interval $[0,1]$. By applying the initial condition given in Equation (16), we obtain

$$-V_s(Y) = \sum_{n=1}^{\infty} A_n \sin \lambda_n Y \tag{36}$$

Equation (36) represents the expansion of an arbitrary function $(-V_s(Y))$ in a Fourier series, named after the scientist Fourier who studied such expansions a long time ago. Fourier series do not necessarily have to be expansions in trigonometric functions, and you can learn more about them from Weinberger [1]. The most important aspect of such an expansion is that the set of functions $\{\sin \lambda_n Y\}$ is orthogonal in the interval $[0,1]$. That is

$$\int_0^1 \sin \lambda_m Y \sin \lambda_n Y \; dY = 0, \qquad m \neq n \tag{37}$$

Of course, when $m = n$, the integral is not zero, but is given by

$$\int_0^1 \sin^2 \lambda_n Y \; dY = \frac{1}{2} \tag{38}$$

Therefore, we can use the following recipe for calculating the expansion coefficients A_n. Multiply both sides of Equation (36) by $\sin \lambda_m Y$ where m is a specific integer, and integrate from $Y = 0$ to 1. Every term in the infinite series will reduce to zero because of Equation (37), with the exception of the term that involves an integral that is of the form of Equation (38) with the index n replaced by m. As a result, we obtain

$$\int_0^1 \left(-V_s\left(Y\right)\right) \sin \lambda_m Y \, dY = A_m \int_0^1 \sin^2 \lambda_n Y \, dY = \frac{A_m}{2} \tag{39}$$

so that we can write

$$A_n = 2 \int_0^1 \left(-V_s\left(Y\right)\right) \sin \lambda_n Y \, dY \tag{40}$$

where we have replaced the index m, which is just a placeholder, with the index n. When the result for $V_s\left(Y\right)$ given in Equation (13) is used and the integration is performed, we ultimately find

$$A_n = -\frac{2}{n\pi} \tag{41}$$

The final result for $V\left(T,Y\right)$ can be written as follows.

$$V\left(T,Y\right) = 1 - Y - \frac{2}{\pi} \sum_{n=1}^{\infty} \frac{\exp\left[-n^2 \pi^2 T\right]}{n} \sin n\pi Y \tag{42}$$

We can infer how long it will take to achieve steady state. Of course, the correct answer is infinite time because the exponential functions in the infinite series are never quite zero approaching that value only when $T \to \infty$. But as a practical matter, we can see that when $T = 1$, which corresponds to physical time $t = b^2 / \nu$, the contribution from the infinite series will be negligible, except in a situation where we wish to be extremely precise. We refer to this "time scale" b^2 / ν as the time it takes for momentum to diffuse a distance b. Analogous time scales can be defined for diffusion of thermal energy or diffusion of species by replacing the kinematic viscosity by the thermal diffusivity or the mass diffusivity, respectively.

Summary

Here is a brief summary of the method of "Separation of Variables." It may be used to find solutions of linear partial differential equations. After identifying the governing partial differential equation and the initial and boundary conditions for our physical system, we

1. scaled the problem by using suitable reference quantities;

2. found a solution of the steady-state problem;

3. expressed the solution of the original problem as the sum of the steady-state solution and a transient contribution, in that process formulating a partial differential equation and the initial and boundary conditions for the transient contribution;

4. found a solution of the transient problem by assuming a product form for that solution;

5. invoked the principle of superposition to express the general solution of the transient problem as an infinite sum;

6. used the orthogonality of the basis functions (sines in our problem) to obtain the coefficients that appear in the general transient solution;

7. wrote the complete solution as the sum of the steady and transient solutions.

The method of "Separation of Variables" also can be used to find the solution of other linear problems such as steady-state multi-dimensional conduction or diffusion problems. In such a case, we would not have an initial condition, but there would be more boundary conditions.

Concluding Remarks

If the process for finding the expansion coefficients A_n reminds you of the process we use for expanding spatial vectors in an orthogonal basis set, the resemblance is not superficial. The idea of geometrical orthogonality, which we can visualize in three-dimensional space, is extended to an infinite-dimensional "function space" in developing a basis set for expanding functions. The dot product that we use with spatial vectors is generalized to the "inner product" which is defined as the integral over the interval that we used, for example, in Equation (37). Just as the eigenvectors of a real symmetric tensor can be used to generate an orthogonal set of basis vectors, a certain type of differential operator, called a self-adjoint operator, is used to generate a basis set of "orthogonal eigenfunctions" in the context of expanding arbitrary functions. You can learn more about such ideas from Greenberg [2].

It is worthy of note that the problem of unsteady heat conduction in a solid slab (or a quiescent liquid layer) of thickness b when the temperature at the surface $y = b$ is maintained at the same value that it is initially everywhere in the slab, while the temperature at the surface $y = 0$ is changed to a new value, is described by the same governing equations and boundary conditions in scaled form. The assumptions are that there are no sources or sinks, heat transport occurs only by conduction with a constant thermal conductivity, the density and specific heat of the material are constant, and that the slab is very long and very wide so that end effects and edge effects can be neglected. By analogy, it can be seen that the same equations also describe unsteady diffusion in a similar situation. Other boundary conditions are possible in these problems. For example, one can prescribe the heat or mass flux at a boundary instead of prescribing the temperature, or write the flux at a boundary as being proportional to the temperature difference between the surface and

a constant ambient temperature. All of these cases can continue to be handled by the same solution method, which gives you some idea about the versatility of the mathematical technique in the case of this type of partial differential equation and boundary conditions. As noted in the summary, the method also can be used with other types of linear partial differential equations such as the Laplace equation or the convective diffusion equation that arise in heat or mass transport.

Acknowledgment

I am indebted to Professor John B. McLaughlin of Clarkson University who read a draft of this module and provided suggestions for improvement.

References

1. H.F. Weinberger, A First Course in Partial Differential Equations, Xerox College Publishing, 1965.

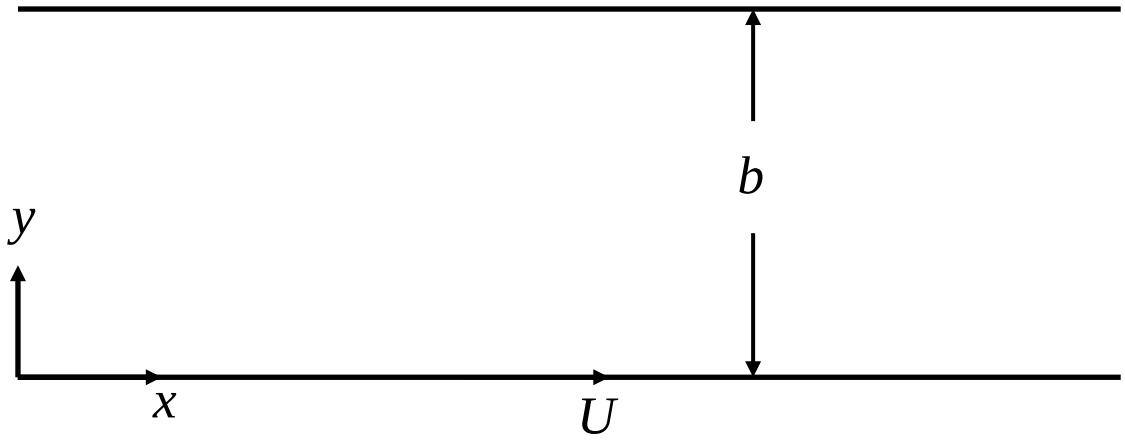
2. M.D. Greenberg, Foundations of Applied Mathematics, Prentice-Hall, 1978.

Solution of Partial Differential Equations

Combination of Variables

Introduction and Problem Statement

We encounter partial differential equations routinely in transport phenomena. Some examples are unsteady flow in a channel, steady heat transfer to a fluid flowing through a pipe, and mass transport to a falling liquid film. Here, we shall learn a method for solving partial differential equations that complements the technique of separation of variables. We shall also learn when the method can be used. We consider the same model problem, namely the motion induced in fluid contained between two long and wide parallel plates placed with a distance b between them as shown in the sketch below.



The fluid is initially assumed to be at rest. Motion is initiated by suddenly moving the bottom plate at a constant velocity of magnitude U in the $x-$direction. The velocity of the bottom plate is maintained at that value for all future values of time t while the top plate is held fixed in place. There is no applied pressure gradient, with motion being caused strictly by the movement of the bottom plate.

We shall assume the flow to be incompressible with a constant density ρ and Newtonian with a constant viscosity μ. We neglect edge effects in the $z-$direction so that we can set $v_z = 0$ and $\frac{\partial \boldsymbol{v}}{\partial z} = 0$, and assume fully developed flow, implying $\frac{\partial \boldsymbol{v}}{\partial x} = 0$. Here, $\boldsymbol{v}$ stands for the velocity vector, and the subscripts denote components.

It can be established from the continuity equation and the kinematic condition at one of the walls that $v_y = 0$. Therefore, the only non-zero velocity component is $v_x(t, y)$, which can be shown to satisfy the following partial differential equation.

$$\frac{\partial v_x}{\partial t} = \nu \frac{\partial^2 v_x}{\partial y^2} \tag{1}$$

In Equation (1), t represents time, and ν is the kinematic viscosity. The initial condition is

$$v_x(0, y) = 0 \tag{2}$$

and the boundary conditions are

$$v_x(t, 0) = U \tag{3}$$

and

$$v_x(t, b) = 0 \tag{4}$$

Using separation of variables, we obtained a solution of these equations that can be written as follows.

$$\frac{v_x(t, y)}{U} = 1 - \frac{y}{b} - \frac{2}{\pi} \sum_{n=1}^{\infty} \frac{\exp\left[-n^2 \pi^2 \frac{\nu t}{b^2}\right]}{n} \sin\left(\frac{n \pi y}{b}\right) \tag{5}$$

The infinite series in Equation (5) is uniformly convergent for all values of time t. The exponential factor plays a strong role in assuring that the terms decrease rapidly with increasing values of n so that only a few terms are necessary to calculate an accurate value of the velocity at moderate to large values of time, corresponding to the scaled time $\nu t / b^2$ not being too small compared to unity. But, if we attempt to calculate the sum numerically for small values of time ($\nu t / b^2$ small compared with unity) when the exponential factor is not as helpful, we find that a large number of terms needs to be included to obtain a sufficiently accurate answer. Therefore, in this module we seek a solution technique that will permit us to calculate the velocity field accurately without too much labor for small values of time.

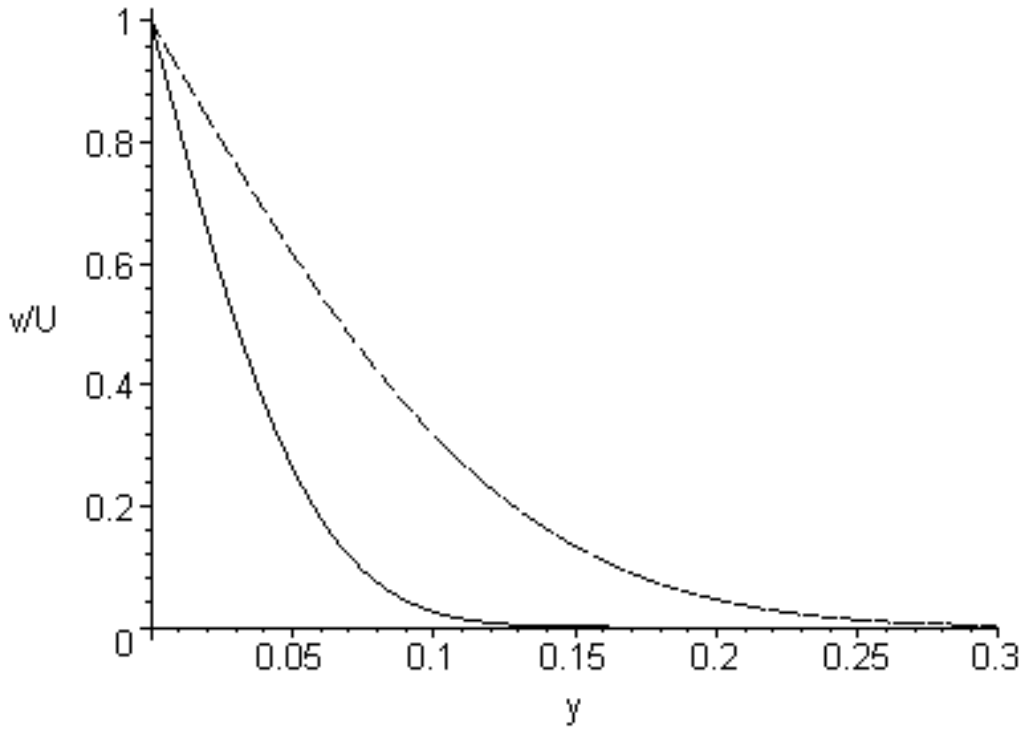
Physically, at values of time t for which the scaled time $\nu t / b^2$ is small compared to unity, the effect of the motion of the bottom plate is only felt by the fluid up to a small distance (depth of penetration) from the moving plate. Outside of this region of influence, the fluid is practically stationary. Therefore, one might approximate the system for such small values of time by another in which the top plate is absent. This problem was first considered by Lord Rayleigh, and therefore is known as Rayleigh's problem. Mathematically, we replace the boundary condition at the top plate, given in Equation (4), with

$$v_x(t, \infty) = 0 \tag{6}$$

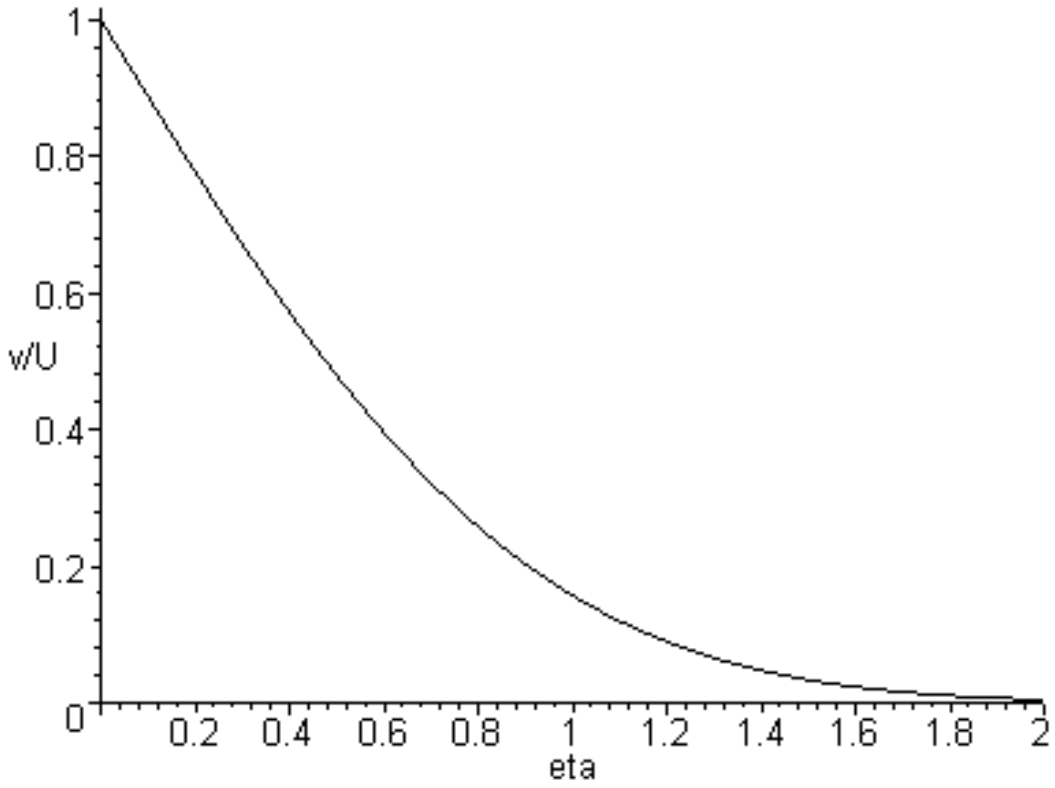
Note that to be precise, we must write Equation (6) as $v_x(t, y \to \infty) \to 0$, and the equation must be read to imply only such a meaning.

A Speculation

There is neither a natural length scale in the problem, nor a natural time scale. We can use the reference velocity U as a natural scale for the velocity v_x, but it is convenient to work with the remaining physical variables just as they are. The solution of Equations (1) to (3) and (6) is qualitatively sketched below for two different values of time. In the figure, the symbol v is used to represent v_x.



The solid line corresponds to a small value of time, and the dashed line to a larger value of time. We can see how the change in velocity made at the bottom plate at time zero propagates deeper into the fluid with increasing time. It is tempting to speculate that these profiles are similar in shape. By implying similarity of shape, we mean that scaling the distance variable with the thickness of the affected region $\delta(t)$ should lead to these two curves and others like them collapsing into a single universal curve. In mathematical language, if we define a certain combination of the original variables as a new variable $\eta = y/\delta(t)$, can we expect the velocity field $v_x(t, y)$ to become a function $U\phi(\eta)$ that depends only on the single new variable? This speculation is shown in the sketch below.



The transformation to η is known as a "similarity transformation" and the variable η is termed a "similarity variable."

Solution by Combination of Variables

We now proceed to state the above speculation in mathematical form and follow through the consequences. This is the method of "Combination of Variables."

Assume

$$v_x(t,y) = U\phi(\eta) \tag{7}$$

where

$$\eta = \frac{y}{\delta(t)} \tag{8}$$

and $\delta(t)$ is a function that is yet to be determined. Note that we always can transform from two independent variables to two new independent variables, but to transform to a single new variable is not always possible. Therefore, we need to insert Equations (7) and (8) into the governing equation and the initial and boundary conditions and see if the process leads to a consistent mathematical framework. For this purpose, we shall use the chain rule of differentiation as needed.

$$\frac{\partial v_x}{\partial t} = U\frac{\partial \eta}{\partial t}\frac{d\phi}{d\eta} = -U\frac{y}{\delta^2}\frac{d\delta}{dt}\frac{d\phi}{d\eta} = -U\frac{\eta}{\delta}\frac{d\delta}{dt}\frac{d\phi}{d\eta} \tag{9}$$

Note that when writing the derivative of ϕ with respect to η, we already have assumed that ϕ can depend explicitly only on the single variable η and used the ordinary derivative instead of the partial derivative. If our conjecture proves to be incorrect, and ϕ were to depend explicitly on both η and t, the above chain rule result will need to be modified to include a partial derivative of ϕ with respect to time.

Let us now obtain expressions for the derivatives with respect to y.

$$\frac{\partial v_x}{\partial y} = U\frac{\partial \eta}{\partial y}\frac{d\phi}{d\eta} = \frac{U}{\delta}\frac{d\phi}{d\eta} \tag{10}$$

and

$$\begin{aligned}\frac{\partial^2 v_x}{\partial y^2} &= \frac{\partial}{\partial y}\left(\frac{\partial v_x}{\partial y}\right) = \frac{\partial}{\partial y}\left(\frac{U}{\delta}\frac{d\phi}{d\eta}\right) = \frac{U}{\delta}\frac{\partial}{\partial y}\left(\frac{d\phi}{d\eta}\right) \\ &= \frac{U}{\delta}\frac{\partial \eta}{\partial y}\frac{d}{d\eta}\left(\frac{d\phi}{d\eta}\right) = \frac{U}{\delta^2}\frac{d^2\phi}{d\eta^2}\end{aligned} \tag{11}$$

Substituting Equations (9) and (11) into the governing differential equation for v_x (Equation (1), leads to the following equation after slight rearrangement.

$$\phi'' + \eta\left[\frac{\delta\delta'}{\nu}\right]\phi' = 0 \tag{12}$$

In writing Equation (12), we have used the expedient of designating derivatives with respect to the argument of each function with primes. Recall that we assumed that ϕ explicitly depends only on the similarity variable η. But, Equation (12) suggests that time also will explicitly appear in the result for ϕ because of the presence of the time-dependent term $\delta\delta'$. We have not yet specified $\delta(t)$, however. Here is our chance to do so and eliminate the inconsistency at the same time. We choose

$$\delta\delta' = C\nu \tag{13}$$

where C is an arbitrary constant. Later, we shall see that the value of C will affect the result for $\delta(t)$, but will not affect the final solution for $v_x(t, y)$. Therefore, for convenience, we set $C = 2$, writing Equation (12) as

$$\phi'' + 2\eta\phi' = 0 \tag{14}$$

We now need to transform the initial and boundary conditions. Note that there are three conditions on the velocity field $v_x(t,y)$, but only a second order differential equation for $\phi(\eta)$. The specification of the arbitrary constants that arise in the integration of the latter requires only two conditions.

First, consider the boundary condition at the bottom surface $y=0$, given in Equation (3). This transforms in a straightforward manner to

$$\phi(0)=1 \tag{15}$$

The fact that a quiescent condition is approached as $y\to\infty$, described by Equation (6), becomes

$$\phi(\infty)=0 \tag{16}$$

The initial condition, given in Equation (2), transforms to

$$\phi\left(\frac{y}{\delta(0)}\right)=0 \tag{17}$$

and we see that we have not completely eliminated the original variables from appearing explicitly in the problem statement for ϕ. To remove this inconsistency, and at the same time select an initial condition for $\delta(t)$, we must set

$$\delta(0)=0 \tag{18}$$

The choice in Equation (18) makes Equation (17) collapse into Equation (16); therefore, the three conditions on $v_x(t,y)$ yield two conditions on $\phi(\eta)$ and one initial condition on $\delta(t)$, and we have a completely consistent mathematical framework for the problems for $\phi(\eta)$ and $\delta(t)$. Note that by this approach of "Combination of Variables" we have reduced the solution of the original partial differential equation to that of two ordinary differential equations for these two functions.

First, the general solution of Equation (14) can be written as

$$\phi(\eta)=a_1+a_2\int_0^{\eta}e^{-\gamma^2}d\gamma \tag{19}$$

where a_1 and a_2 are constants of integration that must be determined by applying the boundary conditions given in Equations (15) and (16). Use of these conditions leads to the result

$$\phi(\eta)=\operatorname{erfc}(\eta) \tag{20}$$

where " erfc " means "complementary error function." This function is defined as follows.

$$\text{erfc}(\eta) = 1 - \text{erf}(\eta) \tag{21}$$

where the "error function" " erf " is defined as

$$erf(\eta) = \frac{\int_0^{\eta} e^{-\gamma^2} d\gamma}{\int_0^{\infty} e^{-\gamma^2} d\gamma} = \frac{2}{\sqrt{\pi}} \int_0^{\eta} e^{-\gamma^2} d\gamma \tag{22}$$

You can find out more about the error function and the complementary error function from Abramowitz and Stegun [1].

The solution of Equation (13) with the constant $C = 2$, when specialized using the initial condition given in Equation (18), is

$$\delta(t) = 2\sqrt{\nu t} \tag{23}$$

When this result for $\delta(t)$ is used in Equation (8) in which η is defined, the solution for the velocity field can be written as

$$v_x(t, y) = U \,\text{erfc}\left(\frac{y}{2\sqrt{\nu t}}\right) \tag{24}$$

If we had made a different choice of value for the constant C that appears in Equation (13), it would have affected the results as follows.

$$\delta(t) = \sqrt{2C\nu t} = \sqrt{\frac{C}{2}}\, 2\sqrt{\nu t} \tag{25}$$

$$\phi(\eta) = erfc\left(\sqrt{\frac{C}{2}}\,\eta\right) \tag{26}$$

You can see that when the definition of η given in Equation (8) is used in Equation (26), along with $\delta(t)$ from Equation (25), the factor $\sqrt{C/2}$ cancels out, leading to the same result for the velocity field given in Equation (24). You may wonder about the uncertainty in the value of $\delta(t)$, which is the thickness of the "affected region," caused by the indeterminacy of the value of C. This is perfectly natural because in a diffusive process, the influence of a change is felt everywhere in the fluid instantaneously. This means that there can be no unambiguous definition of a finite thickness for the affected region; only its scaling can be established uniquely. The complementary error function assumes a value of 4.678×10^{-3} when its argument is 2. Therefore, at a distance

$y = 4\sqrt{\nu t}$, the velocity would be less than 0.5% of the value at the surface of the moving plate, and can be considered negligible for practical purposes. Because of this, the estimate $\left(4\sqrt{\nu t}\right)$ is sometimes used for the thickness of the region influenced by the sudden movement initiated at the boundary.

Summary

In this module, we have learned the method of combination of variables for solving partial differential equations; it complements the method of separation of variables. First, we identified the governing partial differential equation and boundary conditions for our system. Then we

1. noted that the effect of a boundary condition imposed at time zero is felt in a region near that boundary that is small in extent for small values of time and used this fact to replace the boundary condition at the other boundary with one at infinity;

2. assumed that the dependence of the velocity field on the two independent variables can be expressed as a dependence on a single new similarity variable;

3. traced the consequences of this similarity hypothesis mathematically, requiring that the original independent variables not be allowed to appear explicitly in the problem posed in the new similarity variable;

4. obtained an ordinary differential equation for the thickness of the affected region and another ordinary differential equation for the velocity field;

5. collapsed the three boundary conditions on the velocity field into two on the velocity field as expressed in the similarity variable, also yielding an initial condition for the thickness of the affected region;

6. solved these ordinary differential equations to obtain results for the thickness of the affected region and the velocity field;

7. noted that the thickness of the affected region can only be defined to within a multiplicative arbitrary constant, whereas the velocity field is uniquely determined.

The important features of the method are that the domain must be semi-infinite, and the boundary condition at infinity must be the same as the initial condition; even though the problem we posed is linear, the method is equally applicable to non-linear problems.

Concluding Remarks

The problem of unsteady one-dimensional heat conduction in a semi-infinite solid slab (or a quiescent liquid layer) in the $y-$direction, when the temperature at the surface $y = 0$ is changed to a new value at time zero, is described by the same governing equations and boundary conditions. The assumptions are that there are no sources or sinks, heat transport occurs only by conduction

with a constant thermal conductivity, the density and specific heat of the material are constant, and that the slab is very long and very wide so that end effects and edge effects can be neglected. By analogy, it can be seen that the same equations also describe unsteady diffusion in a similar situation. All of these cases can be handled by the same solution method. Note that unlike separation of variables, combination of variables does not require the system of governing equation and boundary conditions to be linear. This method has used successfully in solving the Navier-Stokes equations including inertia (and therefore non-linear) in forced boundary layer flows, and also in solving problems of natural convection in boundary layers wherein the fluid mechanics and heat transport problems lead to coupled non-linear governing equations.

Reference

1. M. Abramowitz and I. A. Stegun, Handbook of Mathematical Functions, Dover, 1965.

Part V

Scaling of Navier-Stokes Equation
Reynolds Number
Laminar Boundary Layer Theory

Scaling of the Navier-Stokes Equation

The purpose of scaling, or non-dimensionalization, of a problem is to identify the true parameters that influence the situation. One might think that these parameters are already known – physical properties such as the density and the viscosity, geometrical parameters such as the diameter of an object or the width of a flow channel, and a typical velocity that characterizes the flow. In reality, because the Navier-Stokes equation is concerned with a balance of forces, the true parameters that influence the behavior of solutions of the Navier-Stokes equation have no dimensions, and are ratios of characteristic forces in the system. By scaling the Navier-Stokes equation, we identify them. In the process, we also minimize the number of parameters. You might have learnt from earlier courses that if a problem has M parameters and N dimensions (such as length, mass, and time), then the number of dimensionless parameters is M-N. In the following, for simplicity, we assume the density ρ to be constant.

We begin with the Navier-Stokes equation

$$\rho\left(\frac{\partial \boldsymbol{v}}{\partial t} + (\boldsymbol{v} \bullet \nabla)\boldsymbol{v}\right) = -\nabla p + \mu\nabla^2 \boldsymbol{v}$$

where p is the hydrodynamic pressure. By using this form, we have restricted consideration only to problems in which gravity does not play any role beyond producing a variation of pressure with depth.

The first step in scaling is to choose reference quantities. To be meaningful, these must be characteristic (or typical) entities that would prevail in the flow.

Reference velocity: v_0; Reference length: D; Reference pressure difference: p_0

For scaling pressure, we usually subtract a datum p_d and divide by p_0, which is established by consideration of whether viscous or inertial effects dominate the flow. For scaling time t, at this stage let us just use a reference time t_0, and choose its definition later.

As an example, for pressure driven flow through a tube, we may choose the reference velocity as the average or maximum velocity in the flow, and the reference length as the tube diameter or radius. One possible reference pressure difference would be the pressure drop over the length of the tube, but we can make other choices as well. For the problem of a sphere settling in a fluid, we might choose the sphere diameter as the reference length, and its settling velocity as the reference velocity.

By defining scaled quantities $\boldsymbol{v}^* = \frac{\boldsymbol{v}}{v_0}$, $t^* = \frac{t}{t_0}$, and $\nabla^* = D\nabla$, we obtain the following scaled version of the Navier-Stokes equation, in which we have used the kinematic viscosity $\nu = \mu / \rho$ in some of the coefficients.

$$\frac{D^2}{\nu t_0}\frac{\partial \boldsymbol{v}^*}{\partial t^*} + \frac{Dv_0}{\nu}\left[\left(\boldsymbol{v}^* \bullet \nabla^*\right)\boldsymbol{v}^*\right] = -\frac{Dp_0}{\mu v_0}\nabla^* p^* + \nabla^{*2}\boldsymbol{v}^*$$

We can make different choices for the scale p_0. In a flow dominated by viscous forces, a logical choice would be a viscous scale $p_0 = \mu v_0 / D$ so that

$$p^* = \frac{p - p_d}{\mu v_0 / D}$$

where p_d is a constant datum from which pressure is measured. Substituting for p^* in the right side of the scaled Navier-Stokes equation, the first term becomes $-\nabla^* p^*$. On the other hand, if a flow situation is dominated by inertia, a more appropriate scale for pressure differences would be an inertial scale $p_0 = \rho v_0^2$. In this case,

$$p^* = \frac{p - p_d}{\rho v_0^2}$$

When this definition of scaled pressure is used in the first term in the right side of the scaled Navier-Stokes equation, the term becomes

$$-\frac{Dv_0}{\nu}\nabla^* p^*.$$

Regarding the reference time t_0, it is possible to make two different choices. If we make the choice $t_0 = D^2 / \nu$, then, the unsteady term in the scaled Navier-Stokes equation is multiplied by unity, while the convective acceleration term $\left[\left(\boldsymbol{v}^* \bullet \nabla^*\right)\boldsymbol{v}^*\right]$ is multiplied by the dimensionless group Dv_0 / ν. On the other hand, it is also possible to examine the flow phenomena on a time scale $t_0 = D / v_0$, in which case the unsteady term in the scaled Navier-Stokes equation is multiplied by the same group Dv_0 / ν that also multiplies the convective acceleration term. This distinction in the choice of a time scale is useful in unsteady problems in which the group Dv_0 / ν is negligibly small. It is a good idea now to investigate the physical meaning of the group Dv_0 / ν, known as the Reynolds number.

Reynolds Number

We define the Reynolds Number of the flow as

$$\mathrm{Re} = \frac{Dv_0\rho}{\mu} = \frac{Dv_0}{\nu}$$

where $\nu = \mu / \rho$ is the kinematic viscosity.

The Reynolds Number multiplies the inertia term when the coefficient of the viscous term is unity. Therefore, it represents the ratio of the importance of the inertia force when compared with the viscous force. Another way to see this is to write the definition of the Reynolds Number slightly differently.

$$\mathrm{Re} = \frac{Dv_0\rho}{\mu} = \frac{\rho v_0^2}{\mu \frac{v_0}{D}} = \frac{\text{inertia force}}{\text{viscous force}}$$

An equivalent physical interpretation is

$$\mathrm{Re} = \frac{\text{Rate of Convective Transport of Momentum}}{\text{Rate of Molecular Transport of Momentum}}$$

The Reynolds Number plays a central role in fluid mechanics. It is important in establishing whether two flow situations are dynamically similar, and in determining conditions for the transition to turbulence. Its magnitude in a given situation tells us whether inertia or viscous forces dominate the flow, or whether both are important.

Some additional considerations

Note that the specific choice of characteristic scales is not critical. For example, for flow through a tube, we can choose either the radius or diameter of the tube for the length scale that characterizes the flow, and the average or maximum velocity as the velocity scale. The numerical value of the Reynolds number will change depending upon the specific choice, but so long as the definitions are consistent, we can establish dynamic similarity between two flows by arranging for the Reynolds number to be the same in both cases. Can you explain why the height of a classroom would not be a good choice for a characteristic length scale for flow around a settling dust particle in the atmosphere, just as the speed of an automobile on the street would not be a good choice for a characteristic velocity scale for flow through a tube, even though from the perspective of forming dimensionless quantities, these are perfectly acceptable choices? A slightly more difficult question is to reason out why the length of a tube is not as good a choice as the diameter of the tube for flow through that tube.

In the beginning, we stated that scaling allows us to establish the dimensionless ratios of characteristic forces in a flow. Yet, we only identified a single dimensionless parameter, namely the Reynolds number, upon scaling the Navier-Stokes equation. Other dimensionless groups emerge as we include additional considerations. For example, if we had retained the gradient of the pressure and the gravitational force separately, we should have obtained another dimensionless group from the Navier-Stokes equation. It is the ratio of the Reynolds number to the Froude number Fr, defined as follows.

$$\mathrm{Fr} = \frac{v_0^2}{Dg}$$

In flow situations not involving free surfaces, the only contribution of the body force term $\rho \boldsymbol{g}$ in the Navier-Stokes equation is to cause an increase of pressure with distance in the direction of the gravity vector. Because such a variation of pressure with height already occurs in a static fluid, it makes no contribution to the flow situation. This is the reason why we started with the Navier-Stokes equation using the dynamic pressure, gradients in which are directly related to the flow. But when free surfaces are present, such as for flow in a river, or flow in a partially filled pipeline, the Froude number can play an important role. In addition, by consideration of the boundary conditions at a free surface, it can be shown that another important dimensionless group, known as the Capillary number, Ca , is important in determining the shape of the interface when viscous forces dominate.

$$Ca = \frac{\mu v_0}{\sigma} = \frac{\mu v_0 / D}{\sigma / D} = \frac{\text{viscous force}}{\text{surface tension force}}$$

The numerator is a measure of the viscous force tending to deform the surface (the associated deforming pressure force also scales in the same way in flow dominated by viscous forces) and the denominator is a measure of the surface tension force that tends to prevent the surface from deforming. When the flow is dominated by inertia, a more appropriate choice is the Weber number, We = $\rho D v_0^2 / \sigma$, which is the product of the Capillary number and the Reynolds number. The Weber number is the ratio of the deforming pressure force (from inertia) to the surface tension force.

Flow at Low Reynolds Number

When the Reynolds number is small, the flow is dominated by viscous forces, with inertia playing a relatively unimportant role. This can happen because of small length scales, small flow velocities, or large kinematic viscosity. Such flows are important in the context of the motion of bacteria and cells in physiological systems, in the behavior of colloidal particles suspended in a fluid, and in microfluidics applications. Flows in which inertia is completely neglected are known as Stokes flows.

By completely neglecting inertia, we obtain the Stokes equation

$$\nabla^2 \boldsymbol{v} = \nabla p$$

which, along with the incompressible equation of continuity

$$\nabla \bullet \boldsymbol{v} = 0$$

forms the set of governing equations. These equations are linear. Therefore, the powerful principle of superposition can be used in solving them. This makes numerous problems amenable to analysis. You will find useful solutions in books by Happel and Brenner (1), Leal (2) and Kim and Karrila (3).

Stokes flows (also known as creeping flows) are reversible in time, from which fact some useful information can be extracted about a situation without solving the problem in detail. As an example, the drag on mirror image objects moving at the same velocity through a fluid in Stokes flow must be of the same magnitude. This is because the simple substitution of $-\boldsymbol{v}$ in place of $\boldsymbol{v}$ (which implies a flow in the reverse direction over the original object) leads to a solution with the dynamic pressure field being given by $-p$ where p is the dynamic pressure field in the original problem. For another example, consider a sphere that experiences a force parallel to a neighboring planar solid surface; it cannot drift toward or away from that planar surface, and must move parallel to it. If it drifts one way or the other in the lateral direction normal to the planar surface, time-reversing the motion of the sphere would lead to a drift in the opposite direction. That cannot be correct, because symmetry dictates that if the sphere moves toward the planar surface in one case, it must also move toward it in the second case. Thus, to satisfy symmetry and reversibility, a sphere moving parallel to a planar surface cannot drift laterally. This argument can be extended to objects that are symmetrical in shape about a plane that is oriented normal to the planar surface in question.

Also, it can be proved that, of all the possible velocity fields that satisfy the same boundary conditions in a given domain, the dissipation is minimum for Stokes flow. One consequence, for example, is that the drag on an object moving through a fluid at a given velocity is larger when including inertia effects than when they are neglected.

Flow at Large Reynolds Number

The viscosity of common fluids such as air and water is very small. Therefore, flow in such fluids often occurs at large values of the Reynolds number. In certain types of problems, it is permissible to neglect the relatively small role of viscosity altogether. An example is the formation of waves on a free surface. Such flows are known as inviscid flows. Furthermore, it is possible to show that in a region of inviscid fluid in which the vorticity $\omega = \nabla \times \boldsymbol{v}$ is zero at some time, it will remain zero for all future values of time. A flow in which the vorticity is zero is known as an irrotational flow.

We know that a vector whose curl is zero can be written as the gradient of a scalar field. Therefore, we can write $\boldsymbol{v} = \nabla \phi$ for irrotational flow. In an incompressible flow, the continuity equation reduces to $\nabla \bullet \boldsymbol{v} = 0$. Using this fact, we can see that the velocity potential ϕ satisfies Laplace's equation $\nabla^2 \phi = 0$.

This equation is much easier to solve than the non-linear Navier-Stokes equation. When the potential is known, we can obtain the velocity field from its gradient, and the inviscid Navier-Stokes equation then is used to infer the pressure field, if desired. In the 19th century this approach was used successfully in solving several problems. Unfortunately, potential flows exert no drag on an object moving through a fluid, no matter how large the value of the Reynolds number, in stark contrast with physical reality. The reason is that when the viscous terms are set equal to zero, we lose the ability to satisfy some boundary conditions because the order of the governing equations is reduced. As it happens, the boundary conditions far from an object as well as the kinematic condition at the surface of an object can be satisfied, but the no-slip boundary condition cannot.

In 1903, in an important contribution, Prandtl showed that, no matter how large the value of the Reynolds number might be, one must account for viscous terms in the neighborhood of solid surfaces. He showed how other simplifications can be made so that the Navier-Stokes equations are made amenable to solution.

Elements of Prandtl's Boundary Layer Theory

The failure of potential flow (incompressible irrotational flow) theory to predict drag on objects when a fluid flows past them provided the impetus for Prandtl to put forward a theory of the boundary layer adjacent to a rigid surface. Prandtl's principal assumptions are listed below.

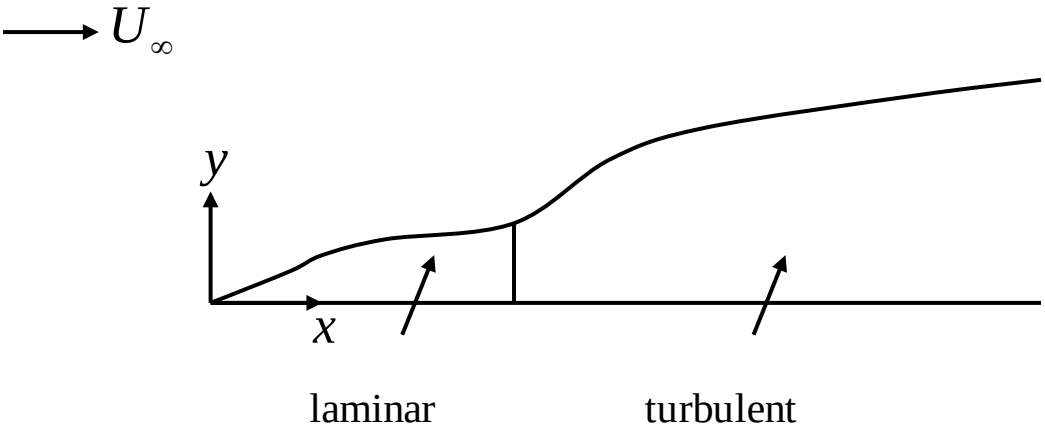
Assumptions

1. When a fluid flows past an object at large values of the Reynolds number, the flow region can be divided into two parts.

(i) Away from the surface of the object, viscous effects can be considered negligible, and potential flow can be assumed.

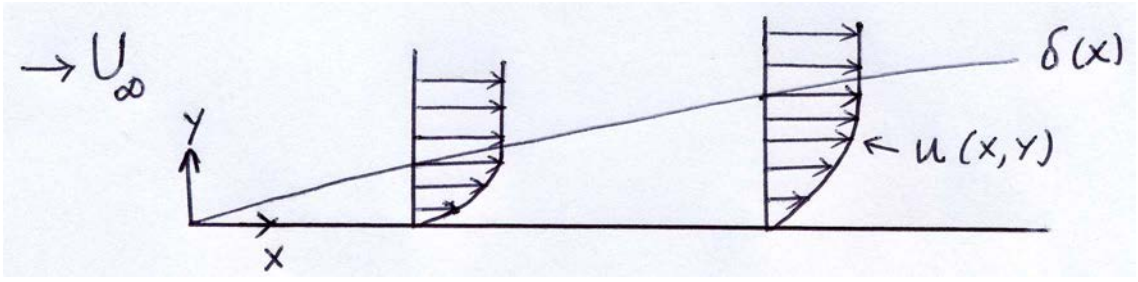
(ii) In a thin region near the surface of the object, called the boundary layer, viscous effects cannot be neglected, and are as important as inertia.

2. The pressure variation can be calculated from the potential flow solution along the surface of the object, neglecting viscous effects altogether, and assumed to be impressed upon the boundary layer.



Transition from laminar to turbulent flow in the boundary layer on a flat plate occurs at $\mathrm{Re}_x \approx 5\times10^5$, where $\mathrm{Re}_x = \dfrac{xU_\infty}{\nu}$. Here, ν is the kinematic viscosity of the fluid.

Qualitative velocity profiles in a laminar boundary layer are displayed below.



The assumptions can be used to establish the order of magnitude of the boundary layer thickness.

A typical inertia term in the Navier-Stokes equation in rectangular Cartesian coordinates is $\rho u\left(\partial u/\partial x\right)$, and a typical viscous term is $\mu\left(\partial^2 u/\partial y^2\right)$. Here, (u,v) are the velocity components in the (x,y) directions, and ρ and μ are the density and the dynamic viscosity of the fluid.

We can estimate the order of magnitude of each of these terms for a plate of length L as follows.

$$\rho u\frac{\partial u}{\partial x}\sim\rho\frac{U_\infty^2}{L} \qquad\qquad \mu\frac{\partial^2 u}{\partial y^2}\sim\mu\frac{U_\infty}{\delta^2}$$

Because the viscous force in the boundary layer is of comparable order to the inertia force, these two order estimates must be comparable.

$\rho\dfrac{U_\infty^2}{L}\sim\mu\dfrac{U_\infty}{\delta^2}$ or $\delta^2\sim\dfrac{\mu L}{\rho U_\infty}$, which can be recast as

$$\frac{\delta}{L}\sim\frac{1}{\sqrt{\mathrm{Re}_L}}$$

where the Reynolds number based on the length of the plate $\mathrm{Re}_L=\dfrac{LU_\infty}{\nu}$.

This type of argument is called a **scaling analysis**. It is a valuable tool in dealing with transport problems. You can see that it provides not only an idea of the variables on which key quantities depend, but also the form of this dependence without having to solve the partial differential equations involved.

In a like manner, we can find a scale estimate of the drag as well. The shear stress at the plate surface is $\tau_w=\mu\dfrac{\partial u}{\partial y}(x,0)$. We can estimate the order of this quantity as $\tau_w(x)=\mu\left(U_\infty/\delta\right)$. Because the shear stress is a local quantity, we should use an order of magnitude of the variation of the boundary layer thickness δ with x. From the order of magnitude argument used earlier, we can estimate it as $\delta(x)\sim x/\sqrt{\mathrm{Re}_x}=\sqrt{\nu x/U_\infty}$. If the width of the plate in the $z-$direction is w, the drag on the plate surface is given by

$$D=w\int_0^L\tau_w(x)\,dx=\mu w\frac{U_\infty^{3/2}}{\nu^{1/2}}\int_0^L\frac{dx}{\sqrt{x}}.$$

Ignoring the numerical factor of 2 that appears after performing the integration (because we are only estimating the order of magnitude), we can write

$$D \sim w\sqrt{\rho \mu U_\infty^3 L}$$

A rigorous calculation from boundary layer theory yields the result

$$D = 0.664\, w\sqrt{\rho \mu U_\infty^3 L}$$

confirming the correctness of our scaling argument.

The Displacement Thickness

The displacement thickness of the boundary layer is defined as the distance by which the potential flow streamlines are displaced by the presence of the boundary layer. We can construct a mathematical definition in the case of the flat plate by recognizing that the displacement thickness δ_1 is that thickness of the uniform stream that accounts for the "lost" flow because of the presence of the solid surface.

$$U_\infty \delta_1 = \int_0^\infty (U - u)\, dy$$

or

$$\delta_1 = \int_0^\infty \left(1 - \frac{u}{U_\infty}\right) dy$$

Order of magnitude analysis of the continuity and Navier-Stokes Equations

Now, we shall go through an order of magnitude analysis of the two-dimensional Navier-Stokes equations for steady incompressible Newtonian laminar flow over a flat plate and simplify them using Prandtl's ideas. For more details, you can consult Chapter VII from Schlichting (4).

We shall use scaled variables, using L as a reference length, and U_∞ as a reference velocity. The symbols x and y are used for the scaled counterparts of the physical coordinates in the sketch, and the symbols u and v are used for the dimensionless counterparts of the physical velocity components in the x and y directions, respectively. The scaled incompressible version of the continuity equation is

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

From the scaling, we know that u is $O(1)$. This means that the magnitude of u lies between 0 and a number that is of the order of unity. In this particular case, because the maximum value of the physical velocity is that of the uniform stream approaching the plate, namely U_∞, the maximum value of u is, in fact, precisely, 1. But this is not necessarily the meaning implied by the

order symbol that we are using. Note that the order of magnitude of a quantity is the same regardless of its sign.

Because the velocity u varies in the range mentioned above, while the scaled variable x also varies from 0 to 1 (we say $x \sim O(1)$), we can conclude that the derivative $\partial u / \partial x$ is $O(1)$ as well. From the continuity equation, we see that $\partial u / \partial x$ and $\partial v / \partial y$ must sum to zero; this forces the derivative $\partial v / \partial y$ to be $O(1)$. We know that the variable $y \sim O(\delta)$ where δ represents the boundary layer thickness divided by the length L. In other words, δ is the scaled boundary layer thickness. Because the derivative $\partial v / \partial y \sim O(1)$, we must conclude that the change in the scaled velocity component v across the boundary layer must be of $O(\delta)$. We know from the kinematic condition that $v = 0$ at the surface $y = 0$. Therefore, the magnitude of v must of $O(\delta)$. We note that δ is a very small quantity when the Reynolds number $\mathrm{Re}_L \gg 1$. We express this fact by stating $\delta \ll 1$. Therefore, the scaled velocity in the $y-$direction in the boundary is a very small quantity.

Now, following Schlichting (4) we proceed to use similar arguments in the two components of the Navier-Stokes equation applicable to this situation. First, consider the $x-$component.

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{\mathrm{Re}_L}\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right]$$

$$1 \quad 1 \qquad \delta \quad \frac{1}{\delta} \qquad\qquad\qquad \delta^2 \quad 1 \qquad \frac{1}{\delta^2}$$

Below each term in the equation, we have written the order of magnitude of that term. We already have discussed the order of magnitude of u, v, and $\partial u / \partial x$. To estimate the order of magnitude of $\partial u / \partial y$, we first note that u varies from 0 to 1 across the boundary layer, while the variable y varies from 0 to δ. This is the reason for the estimate that $\partial u / \partial y \sim O(1/\delta)$. To estimate the order of magnitude of the second derivatives, we must use similar arguments. For example, consider the derivative $\partial^2 u / \partial x^2$. We know that $\partial u / \partial x \sim O(1)$. So, this quantity must change from 0 to a magnitude of the order of unity in a scaled distance x that also changes from 0 to 1. This is the reason for estimating the order of $\partial^2 u / \partial x^2$ as being unity. In a like manner, the derivative $\partial u / \partial y \sim O(1/\delta)$, which means that it varies from 0 to $1/\delta$ across the boundary layer, in a distance of the order δ. Therefore, the second derivative $\partial^2 u / \partial y^2 \sim O(1/\delta^2)$. The order of magnitude of the Reynolds number was established earlier.

Comparing the two viscous terms, we see that the viscous force in the $x-$direction is negligible when compared to that in the $y-$direction. We need to retain all the other terms in the $x-$component momentum equation because they are all of comparable order of magnitude.

Now, let us consider the $y-$component of the Navier-Stokes equation.

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{\mathrm{Re}_L}\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right]$$

$$1 \quad \delta \qquad \delta \quad 1 \qquad\qquad\qquad \delta^2 \quad \delta \qquad \frac{1}{\delta}$$

The order of magnitude of the derivatives has been estimated in the same manner as outlined earlier. Once again, we see that the viscous transport of $y-$momentum in the $x-$direction is much weaker than that in the $y-$direction, and can be neglected. The most important aspect of the above equation is that all the retained terms are of $O(\delta)$, so that the pressure gradient $\partial p/\partial y$ must necessarily be of the same order (or smaller). Because the variation of pressure in the $y-$direction in the boundary layer must occur over a distance of $O(\delta)$, it is evident that the scaled pressure change across the thickness of the boundary layer $\Delta p \sim O(\delta^2)$. This is very small, and can be ignored, which is Prandtl's assumption 2. Because the pressure change across the boundary layer is negligible, the pressure distribution along the surface of the object, evaluated from the potential flow, can be assumed to be "impressed" on the boundary layer. This means that $\partial p/\partial x$ in the $x-$component momentum equation is a known inhomogeneity, and we can simply ignore the $y-$component momentum equation because all the terms are small.

Summarizing the above, we have found from the scaling analysis that the viscous term in the main direction of flow (x) is negligible compared with the viscous term in the direction normal to the solid surface. Furthermore, the pressure gradient in the $x-$component momentum equation is established from potential flow theory and evaluated along the surface of the object, and the $y-$component momentum equation is neglected. Thus, we have two equations for the two unknown velocity components.

Even though our analysis assumed a flat plate, you can see that for a thin boundary layer, the effects of curvature of the surface would be negligible at leading order. Therefore, as long as we define x and y as distance coordinates along and normal to a surface, respectively, the same equations can be written for flow past an object with a curved surface. For convenience, Prandtl's steady two-dimensional boundary layer equations for incompressible Newtonian flow are written in physical variables below. To avoid clutter, we have retained the same symbols for the velocities and coordinates as those used earlier for scaled variables, but this should not be a source of confusion.

Continuity

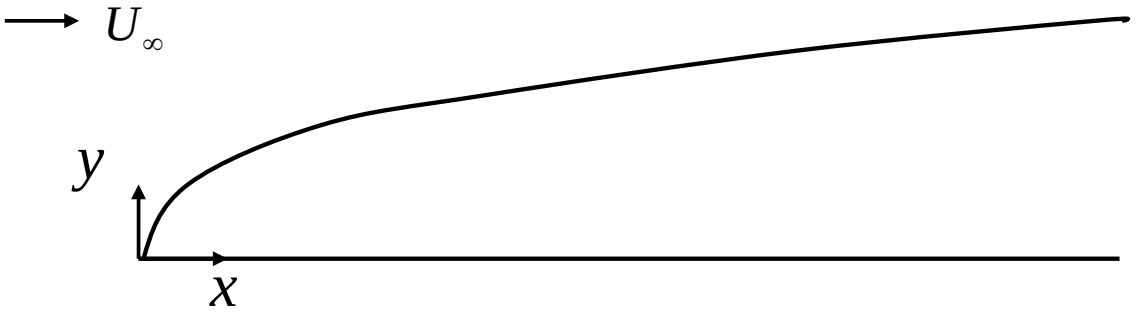
$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Navier-Stokes Equation

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\frac{\partial^2 u}{\partial y^2}$$

The simplest boundary layer theory problem is that of steady two-dimensional laminar incompressible Newtonian flow over a flat plate. The term "two-dimensional" implies that the plate is wide in the $z-$direction normal to the plane of the paper.

Boundary layer on a flat plate



For the flat plate problem, the potential flow is simply $u = U_\infty$. This means that the potential flow pressure gradient is zero. Therefore, the Navier-Stokes equation simplifies to

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = \nu\frac{\partial^2 u}{\partial y^2}$$

The boundary conditions are written as follows.

$u(0, y) = U_\infty$ Specified uniform flow at $x = 0$

$u(x, 0) = 0$ No slip at the solid surface

$v(x, 0) = 0$ Kinematic condition at the solid surface

$u(x, y \to \infty) \to U_\infty$ Free stream velocity as $y \to \infty$

Commonly, the last condition is replaced with $u(x, \infty) = U_\infty$.

Asymptotic Analysis of Navier-Stokes Equation for Large Reynolds Number

The simplified versions of the continuity and Navier-Stokes equations that we obtained for boundary layer flow over a flat plate using an order of magnitude analysis also can be obtained by a rigorous analysis of the governing equations in the asymptotic limit $\mathrm{Re}_L \to \infty$, where we define the Reynolds number as $\mathrm{Re}_L = LU_\infty / \nu$. Here, L is a reference length chosen as the length of the flat plate, the reference velocity U_∞ is the uniform free stream velocity in the $x-$ direction in the fluid approaching the flat plate, and ν is the kinematic viscosity of the fluid. There are several good sources in which such an asymptotic analysis can be found. The derivation given below is adapted from that of Leal (2). We shall use the same scaled continuity and Navier-Stokes equation components that were used in the last section.

Continuity

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

Navier-Stokes Equations

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{\mathrm{Re}_L}\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right]$$

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{\mathrm{Re}_L}\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right]$$

We know that in the limit as $\mathrm{Re}_L \to \infty$, the viscous terms become vanishingly small, yielding the simple potential flow solution $u = U_\infty, v = 0$, and the pressure is uniform, so that $\partial p / \partial x = 0$. This potential flow solution satisfies the continuity and Navier-Stokes equations in the limit $\mathrm{Re}_L \to \infty$, the boundary condition as $y \to \infty$, and the kinematic boundary condition $v(x,0) = 0$ at the solid surface. But, this solution does not satisfy the boundary condition of no slip at the solid surface, namely $u(x,0) = 0$, and therefore is not uniformly valid everywhere. This failure is a direct result of the neglect of the viscous terms in the Navier-Stokes equation. The method that is used to accommodate these terms is known as matched asymptotic expansions, and you can learn about this method from books by Van Dyke (5), Nayfeh (6) or Kevorkian and Cole (7). Here, we shall use the method in its simplest form applicable to the current situation. The main idea is that, in order to retain the relevant viscous term, we must "magnify" the region near the solid boundary. Another term that is used to describe the process is "rescale" the y-coordinate in the boundary layer. Because the Reynolds number is large, such rescaling is accomplished by postulating a new coordinate $\tilde{y} = \mathrm{Re}_L^\alpha \, y$, where we require the constant exponent $\alpha > 0$. The value of α will be established as part of the analysis. In the following, we shall use the term "leading order" to

designate the first term in an asymptotic expansion in the limit $\mathrm{Re}_L \to \infty$, which is the same as $(1/\mathrm{Re}_L) \to 0$. The derivatives in y will transform as follows:

$$\frac{\partial u}{\partial y} = \frac{d\tilde{y}}{dy}\frac{\partial u}{\partial \tilde{y}} = \mathrm{Re}_L^{\alpha}\frac{\partial u}{\partial \tilde{y}}\ , \quad \frac{\partial^2 u}{\partial y^2} = \frac{\partial}{\partial y}\left(\frac{\partial u}{\partial y}\right) = \frac{d\tilde{y}}{dy}\frac{\partial}{\partial \tilde{y}}\left(\frac{\partial u}{\partial y}\right) = \frac{d\tilde{y}}{dy}\frac{\partial}{\partial \tilde{y}}\left(\mathrm{Re}_L^{\alpha}\frac{\partial u}{\partial \tilde{y}}\right) = \mathrm{Re}_L^{2\alpha}\frac{\partial^2 u}{\partial \tilde{y}^2}$$

$$\frac{\partial v}{\partial y} = \frac{d\tilde{y}}{dy}\frac{\partial v}{\partial \tilde{y}} = \mathrm{Re}_L^{\alpha}\frac{\partial v}{\partial \tilde{y}}\ , \quad \frac{\partial^2 v}{\partial y^2} = \frac{\partial}{\partial y}\left(\frac{\partial v}{\partial y}\right) = \frac{d\tilde{y}}{dy}\frac{\partial}{\partial \tilde{y}}\left(\frac{\partial v}{\partial y}\right) = \frac{d\tilde{y}}{dy}\frac{\partial}{\partial \tilde{y}}\left(\mathrm{Re}_L^{\alpha}\frac{\partial v}{\partial \tilde{y}}\right) = \mathrm{Re}_L^{2\alpha}\frac{\partial^2 v}{\partial \tilde{y}^2}$$

Substituting the result for $\partial v/\partial y$ in the equation of continuity yields the following transformed continuity equation in the new coordinates $(x, \tilde{y})$.

$$\frac{\partial u}{\partial x} + \mathrm{Re}_L^{\alpha}\frac{\partial v}{\partial \tilde{y}} = 0$$

The two terms in the left side need to be of the same order of magnitude in order to add to a value of zero. Because $\mathrm{Re}_L^{\alpha} \to \infty$ as $\mathrm{Re}_L \to \infty$, we must rescale the velocity component in the $y-$ direction in the boundary layer by defining $\tilde{v} = \mathrm{Re}_L^{\alpha} v$. This rescaling transforms the continuity equation in the boundary layer to

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

Now, consider the $x-$component of the Navier-Stokes equation. Transforming to the new $(x, \tilde{y})$ coordinates and using the transformed velocity component $\tilde{v}(x, \tilde{y})$, we obtain the following transformed result.

$$u\frac{\partial u}{\partial x} + \mathrm{Re}_L^{-\alpha}\,\tilde{v}\,\mathrm{Re}_L^{\alpha}\frac{\partial u}{\partial \tilde{y}} = -\frac{\partial p}{\partial x} + \frac{1}{\mathrm{Re}_L}\left[\frac{\partial^2 u}{\partial x^2} + \mathrm{Re}_L^{2\alpha}\frac{\partial^2 u}{\partial \tilde{y}^2}\right]$$

or

$$u\frac{\partial u}{\partial x} + \tilde{v}\frac{\partial u}{\partial \tilde{y}} = -\frac{\partial p}{\partial x} + \mathrm{Re}_L^{-1}\frac{\partial^2 u}{\partial x^2} + \mathrm{Re}_L^{2\alpha-1}\frac{\partial^2 u}{\partial \tilde{y}^2}$$

Considering the two viscous terms in the right side, the critical viscous term that must be retained in the governing differential equation in order to satisfy the no-slip boundary condition is $\left(\mathrm{Re}_L^{2\alpha-1}\right)\partial^2 u/\partial \tilde{y}^2$. Recalling Prandtl's assumption in the boundary layer that this viscous term

must be of the same order of magnitude as the inertia terms, we see that we must choose $\alpha = 1/2$, leading to the following result.

$$u\frac{\partial u}{\partial x} + \tilde{v}\frac{\partial u}{\partial \tilde{y}} = -\frac{\partial p}{\partial x} + \mathrm{Re}_L^{-1}\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial \tilde{y}^2}$$

In this transformed equation, if we take the limit $\mathrm{Re}_L \to \infty$, we obtain the leading order boundary layer equation for the $x-$component of the momentum.

$$\boxed{u\frac{\partial u}{\partial x} + \tilde{v}\frac{\partial u}{\partial \tilde{y}} = -\frac{\partial p}{\partial x} + \frac{\partial^2 u}{\partial \tilde{y}^2}}$$

Likewise, we can transform the Navier-Stokes equation for the $y-$component of momentum. First, let us note that $\frac{\partial v}{\partial y} = \mathrm{Re}_L^{\alpha}\frac{\partial v}{\partial \tilde{y}} = \left(\mathrm{Re}_L^{\alpha}\,\mathrm{Re}_L^{-\alpha}\right)\frac{\partial \tilde{v}}{\partial \tilde{y}} = \frac{\partial \tilde{v}}{\partial \tilde{y}}$. The $y-$component of the momentum equation can be written as follows.

$$\mathrm{Re}_L^{-1/2}\,u\frac{\partial \tilde{v}}{\partial x} + \mathrm{Re}_L^{-1/2}\,\tilde{v}\frac{\partial \tilde{v}}{\partial \tilde{y}} = -\mathrm{Re}_L^{1/2}\frac{\partial p}{\partial \tilde{y}} + \mathrm{Re}_L^{-1}\left[\mathrm{Re}_L^{-1/2}\frac{\partial^2 \tilde{v}}{\partial x^2} + \mathrm{Re}_L^{1/2}\frac{\partial^2 \tilde{v}}{\partial \tilde{y}^2}\right]$$

In the above equation, we already have substituted the value of $\alpha = 1/2$. Multiply both sides of the equation by $\mathrm{Re}_L^{-1/2}$ and rearrange to yield

$$\frac{\partial p}{\partial \tilde{y}} = \mathrm{Re}_L^{-2}\frac{\partial^2 \tilde{v}}{\partial x^2} - \mathrm{Re}_L^{-1}\left[u\frac{\partial \tilde{v}}{\partial x} + \tilde{v}\frac{\partial \tilde{v}}{\partial \tilde{y}} - \frac{\partial^2 \tilde{v}}{\partial \tilde{y}^2}\right]$$

Now, if we take the limit $\mathrm{Re}_L \to \infty$, the leading order boundary layer equation for the $y-$component of the momentum becomes simply

$$\boxed{\frac{\partial p}{\partial \tilde{y}} = 0}$$

Thus, as Prandtl correctly postulated, the variation of pressure within the boundary layer is negligible, and the pressure at the "edge" of the boundary layer from the potential flow is "impressed" on the boundary layer. The pressure gradient that appears in the $x-$component momentum equation is that evaluated from the potential flow at the edge of the boundary layer. However, on the scale of the potential flow, the boundary layer is of negligible thickness. Thus, the pressure gradient from potential flow is actually evaluated at the solid surface $y = 0$, and used in the $x-$component momentum equation. This argument can be made rigorous by using the tools of matched asymptotic expansions.

We see that we have recovered the same continuity and $x-$component momentum equation that we obtained earlier using a simple scaling argument. The no-slip condition, namely $u\left(x,\tilde{y}=0\right)=0$ can be satisfied, because the second derivative in $\tilde{y}$ is now retained in the $x-$component momentum equation. What about a second boundary condition on the velocity field $u\left(x,\tilde{y}\right)$? This is obtained from the "asymptotic matching principle" that is discussed in Van Dyke (5) or Nayfeh (6). Basically, the matching principle requires that as the boundary layer coordinate $\tilde{y}\to\infty$, the boundary layer solution must approach the potential flow solution as the coordinate $y\to 0$. For the case of flow over a flat plate, the potential flow solution is the uniform velocity field $u\left(x,y\right)=U_\infty$ everywhere, so that we must require $u\left(x,\tilde{y}\to\infty\right)\to U_\infty$. We commonly write this condition simply as

$$u\left(x,\infty\right)=U_\infty$$

The continuity equation can be integrated to express the velocity component $\tilde{v}$ in the boundary layer in terms of an integral of $\partial u/\partial x$ as follows.

$$v\left(x,\tilde{y}\right)=v\left(x,0\right)-\int_0^{\tilde{y}}\frac{\partial u}{\partial x}\left(x,\tilde{y}\right)d\tilde{y}$$

and from the kinematic condition at the solid surface the first term on the right side of the above equation is zero. Therefore, we can write

$$\boxed{v\left(x,\tilde{y}\right)=-\int_0^{\tilde{y}}\frac{\partial u}{\partial x}\left(x,\tilde{y}\right)d\tilde{y}}$$

Recall that we have used dimensionless variables in the above analysis. Reverting to physical variables will lead to the same boundary layer equations that were obtained earlier using a scaling analysis.

Summary

In summary, note that

1. The important nonlinear (inertial) terms have been retained.

2. The number of differential equations has been reduced from three to two, consistent with the simplification that the pressure distribution is "known" from potential flow theory.

3. Because the variation of pressure across the boundary layer is negligible to this order of approximation, the potential flow pressure distribution can be evaluated right at the solid surface and used as a known inhomogeneity in the boundary layer equations.

The solution of the boundary layer equations for a flat plate was first obtained by the method of combination of variables (similarity) by Blasius as discussed in Schlichting (4). The calculation of

this solution will be part of a homework assignment. The same equations also can be solved by an approximate technique, which we discuss next.

Approximate method of Karman and Pohlhausen

This method is also known as the integral method. The highlights for the flat plate problem are presented below. There are three main steps.

1. Satisfy the governing equation for the velocity in the $x-$direction only on an average basis across the boundary layer. The boundary layer thickness $\delta(x)$ is assumed to be finite and outside this region, the velocity is assumed to be U_∞. This means that $u(x, y \geq \delta) = U_\infty$. Thus, we shall use the following boundary conditions on $u(x, y)$ when analyzing the boundary layer problem.

$u(0, y) = U_\infty$ Specified uniform flow at $x = 0$

$u(x, 0) = 0$ No slip at the solid surface

$u(x, \delta) = U_\infty$ The free stream velocity is reached at the edge of the boundary layer $y = \delta(x)$,

2. Satisfy the boundary conditions by assuming a suitable velocity profile.

3. Calculate $\delta(x)$ and the Drag D.

For the integral method, we begin with the $x-$component of the Navier-Stokes equation for the boundary layer flow over a flat plate. First, the result for $v(x, y)$ obtained by integrating the continuity equation, and using the kinematic condition $v(x, 0) = 0$, is substituted in this equation to yield

$$u\frac{\partial u}{\partial x} - \left(\int_0^y \frac{\partial u}{\partial x} dy\right)\frac{\partial u}{\partial y} = \frac{\mu}{\rho}\frac{\partial^2 u}{\partial y^2}$$

Step 1: Integrate the boundary layer equation of motion from $y = 0$ to h. The position $y = h$ is assumed to lie outside the boundary layer everywhere, so that $h > \delta(x)$ over the entire length of the plate. Later, we shall let $h \to \infty$.

$$\int_0^h u\frac{\partial u}{\partial x} dy - \int_0^h \left(\int_0^y \frac{\partial u}{\partial x} dy\right)\frac{\partial u}{\partial y} dy = \frac{\mu}{\rho}\left[\frac{\partial u}{\partial y}\right]_0^h = -\frac{\mu}{\rho}\frac{\partial u}{\partial y}(x, 0) = -\frac{\tau_w}{\rho}$$

As you can see, we have integrated the second derivative with respect to y in the right side to yield the first derivative. At the upper limit $y = h$, which is outside the boundary layer, the velocity profile is flat, so that $\frac{\partial u}{\partial y}(x, h) = 0$, which leads to the final result in the right side. The

symbol τ_w represents the tangential (or shear) stress exerted by the fluid on the wall, and is commonly termed the wall shear stress. It is given by

$$\tau_w(x) = \mu\left[\frac{\partial u}{\partial y}(x,0) + \frac{\partial v}{\partial x}(x,0)\right] = \mu\frac{\partial u}{\partial y}(x,0) \text{ because } \frac{\partial v}{\partial x}(x,0) = 0 .$$

The second term in the left side is integrated by parts as follows.

$$-\int_0^h \underbrace{\left(\int_0^y \frac{\partial u}{\partial x}\,dy\right)}_{U} \underbrace{\frac{\partial u}{\partial y}\,dy}_{dV} = -\left[\left(\int_0^y \frac{\partial u}{\partial x}\,dy\right)u\right]_0^h + \int_0^h u\frac{\partial u}{\partial x}\,dy$$

$$= -U_\infty\int_0^h \frac{\partial u}{\partial x}\,dy + \int_0^h u\frac{\partial u}{\partial x}\,dy$$

Substitute into the left side of the integrated boundary layer equation to obtain the following result.

$$\int_0^h (2u - U_\infty)\frac{\partial u}{\partial x}\,dy = -\frac{\tau_w}{\rho} \quad \text{or} \quad \int_0^h \left(\frac{\partial}{\partial x}\left(u^2 - U_\infty u\right)\right)dy = -\frac{\tau_w}{\rho}$$

Rewrite after reversing signs on both sides as

$$\frac{\partial}{\partial x}\int_0^h \left[u\left(U_\infty - u\right)\right]dy = +\frac{\tau_w}{\rho}$$

which can be rearranged as

$$U_\infty^2\frac{\partial}{\partial x}\int_0^h \frac{u}{U_\infty}\left(1 - \frac{u}{U_\infty}\right)dy = \frac{\tau_w}{\rho}$$

In the integral method, $u/U_\infty = 1$ outside the boundary layer, that is for all $y \ge \delta(x)$. Therefore, we can replace the upper limit h in the integral in the left side with ∞, because the addition to the integral due to this replacement is zero. Thus, we obtain the integral momentum balance.

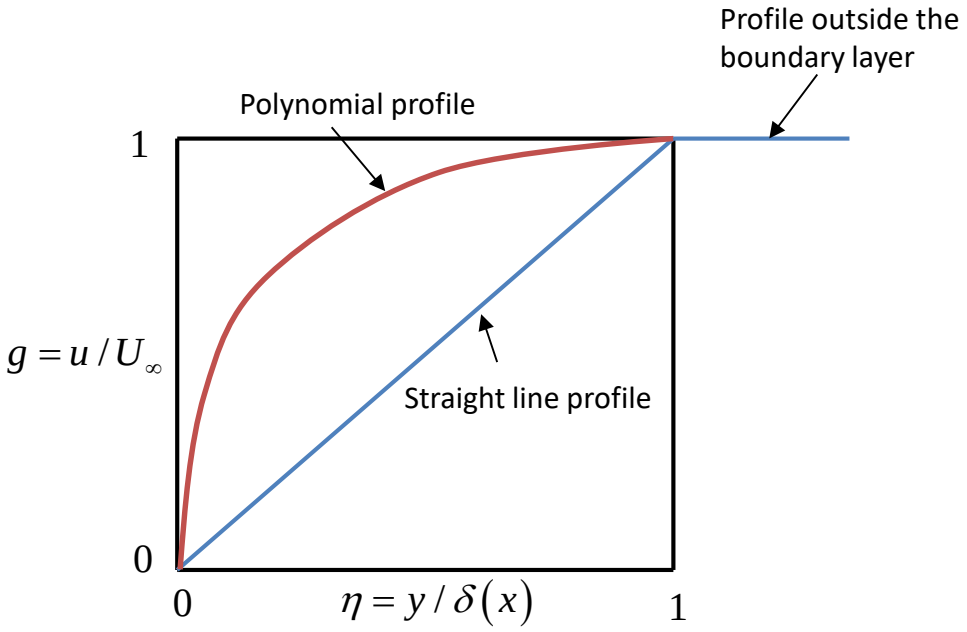
$$U_\infty^2\frac{d\theta}{dx} = \frac{\tau_w}{\rho}$$

Here, the momentum thickness of the momentum boundary layer θ is given by

$$\theta(x) = \int_0^\infty \frac{u}{U_\infty}\left(1 - \frac{u}{U_\infty}\right)dy$$

The physical meaning of the integral momentum balance is simple. It states that the rate at which momentum is lost by the boundary layer fluid is equal to the rate at which it is transferred to the wall.

Step 2: Assume a suitable velocity profile. Because there is no natural length scale in the $y-$ direction, it is reasonable to expect similar profiles at different $x-$locations. Thus, assume that the velocity distribution $u(x,y)=U_\infty g(\eta)$, where $\eta = y/\delta(x)$. Select $g(\eta)$ such that, at a minimum, the boundary conditions on the velocity field $u(x,0)=0$ and $u(x,\delta)=U_\infty$ are satisfied. This implies that $g(0)=0$ and $g(1)=1$. It is common to assume that $g(\eta)$ is a low order polynomial in η. The simplest is a straight line profile $g(\eta)=a_1\eta+a_2$ with two arbitrary constants a_1 and a_2 that can be determined by using the two boundary conditions given above.



For higher order polynomials, we need additional boundary conditions. A linear velocity profile will lead to a discontinuous slope at the edge of the boundary layer, so a third boundary condition one might use for achieving smoothness in the profile is $\frac{\partial u}{\partial y}(x,\delta)=0$. This means that we can set $g'(1)=0$. Higher order polynomials can be assumed, in which the constants are evaluated using additional boundary conditions obtained by evaluating the two sides of the differential equation for $u(x,y)$. For example, if you want to use a fourth order polynomial for $g(\eta)$, two additional conditions obtained from the differential for $u(x,y)$ by evaluation at the surface of the flat plate and at the edge of the boundary layer are

$$\frac{\partial^2 u}{\partial y^2}(x,0)=0;\ \frac{\partial^2 u}{\partial y^2}(x,\delta)=0$$

These two equations therefore lead to the additional conditions $g''(0)=0;\ g''(1)=0$ that can be used to evaluate the constants in the polynomial form assumed for $g(\eta)$.

Step 3: By substituting $u(x,y) = U_\infty g(\eta)$ in the definition of $\theta(x)$, we obtain $\theta(x) = \alpha\,\delta(x)$ where the constant $\alpha = \int_0^1 g(\eta)\left[1 - g(\eta)\right] d\eta$.

Similarly, $\tau_w(x) = \beta \dfrac{\rho \nu U_\infty}{\delta(x)} = \beta \dfrac{\mu U_\infty}{\delta(x)}$ where the constant $\beta = g'(0)$.

Using these results, the integral momentum balance reduces to

$$\frac{d}{dx}\left(\delta^2\right) = \frac{2\beta}{\alpha}\frac{\nu}{U_\infty}$$

and the boundary layer thickness satisfies $\delta(0) = 0$. Integration of this simple ordinary differential equation, and use of the boundary condition on $\delta(x)$ yields

$$\delta(x) = \sqrt{\frac{2\beta}{\alpha}}\sqrt{\frac{\nu x}{U_\infty}}$$

The drag $D(x) = w\int_0^x \tau_w(x)\,dx = w\sqrt{2\alpha\beta}\,\sqrt{\rho\mu U_\infty^3 x}$ where w is the width of the plate. We can define a drag coefficient $C_D = \dfrac{D(x)}{\left(\frac{1}{2}\rho U_\infty^2\right)(wx)} = \dfrac{2D(x)}{\rho U_\infty^2 wx}$. Using the above result for the drag leads to $\boxed{C_D = \dfrac{\sqrt{8\alpha\beta}}{\mathrm{Re}_x^{1/2}}}$

As an example, choose $g(\eta) = a_1\eta + a_2$. Upon using the two boundary conditions $g(0) = 0,\ g(1) = 1$, this yields $g(\eta) = \eta$. Using the definitions of α and β, we obtain $\alpha = \dfrac{1}{6},\ \beta = 1$. Therefore, $\delta(x) = 3.46\sqrt{\nu x / U_\infty}$ and $D = 0.58\, w\sqrt{\rho\mu U_\infty^3 x}$. This leads to $C_D = 1.155 / \mathrm{Re}_x^{1/2}$. This is remarkably close to the correct result from the similarity solution, which is $C_D = 1.328 / \mathrm{Re}_x^{1/2}$. A fourth order polynomial fit to the velocity profile yields $C_D = 1.372 / \mathrm{Re}_x^{1/2}$. More details can be found in Chapter X, Section (a) from Schlichting (4).

The integral method is easy to use, quick, and usually gives the correct scaling results. Only the numerical coefficients are different from those from an exact solution. As the velocity profile is more closely approximated, the estimate of the drag improves. The method can be extended in a straightforward manner to accommodate physical property variations. The integral method also is useful in heat transfer and mass transfer problems involving boundary layers. The principal

disadvantage of this method is that it is not possible to know how accurate the predictions are without comparing either with an exact result, which would obviate the need for the approximate solution, or comparison with experimental results.

Concluding Remarks

1. Boundary layer theory is needed so that we can predict drag in high Reynolds number flow past an object, because potential theory, which is otherwise useful, cannot predict this drag.

2. The theory can be used only up to the point of separation. This is the location on the surface where the velocity gradient in the boundary layer becomes zero at the surface of the object. This topic is further discussed in references (2) and (4).

References

1. J. Happel and H. Brenner, Low Reynolds Number Hydrodynamics, Prentice-Hall, 1965 (reprinted by Noordhoff International, 1973)

2. L.G. Leal, Advanced Transport Phenomena: Fluid Mechanics and Convective Transport Processes, Cambridge University Press, 2007.

3. S. Kim and S.J. Karrila, Microhydrodynamics: Principles and Selected Applications, Butterworth-Heinemann, 1991.

4. H. Schlichting, Boundary Layer Theory, McGraw-Hill, 1968.

5. M. Van Dyke, Perturbation Methods in Fluid Mechanics, Parabolic Press, 1975.

6. A.H. Nayfeh, Perturbation Methods, Wiley, 1973.

7. J. Kevorkian and J.D. Cole, Perturbation Methods in Applied Mathematics, Springer-Verlag, 1981.

Part VI

Introduction to Energy Transport

Conservation of Energy
Boundary Conditions
Heat Transfer Coefficient
Important Dimensionless Groups

Introduction to Energy Transport

Transport of thermal energy in fluids occurs by three mechanisms.

Conduction or molecular transport
Convection or bulk transport
Radiation

Of these, the proper treatment of radiation is beyond the scope of our course. Therefore, we shall only consider conduction and convection.

Conservation of Energy

The energy of a flowing fluid consists of internal and kinetic energy. The rate of increase of the energy content of the fluid present within a control volume at a given instant is equal to the sum of the net flux of energy into the control volume and the work done on the fluid within the control volume by body forces and surface forces acting on it. Bird et al. go through a careful derivation of the mathematical form of the equation of conservation of energy in Chapter 11. The initial balance is written for the total energy and then the part involving the kinetic energy (known as the mechanical energy balance, obtained by taking a dot product of Cauchy's equation with the velocity) is subtracted. After using some thermodynamic relationships, the final form of the equation of conservation of thermal energy is obtained. Various versions are given in Table 11.4-1 of the book. The most common version that we shall use assumes that the density ρ and thermal conductivity k are constant, and is given below.

$$\rho C_p \left(\frac{\partial T}{\partial t} + \boldsymbol{v} \bullet \nabla T \right) = k \nabla^2 T + \mu \, \Phi_v + S$$

In the above equation, T is the temperature, C_p is the specific heat at constant pressure, t is time, $\boldsymbol{v}$ is the (vector) velocity, $\mu \, \Phi_v$ stands for the rate of irreversible conversion of mechanical energy into internal energy per unit volume by viscous dissipation, and S represents the rate of generation of energy per unit volume by sources such as electrical heating. When the thermal energy equation is obtained for a multicomponent system, the rate of generation (or consumption) of energy per unit volume due to chemical reactions appears naturally in the energy equation when proper accounting is made of the enthalpies of the various species. Detailed expressions for the dissipation function Φ_v in terms of derivatives of the velocity components in common coordinate systems can be found in Table B.7 of Bird et al. In most situations, we can set the viscous dissipation term to zero with negligible error, the exceptions occurring when highly viscous fluids are subjected to large velocity gradients; an example where viscous heating is important is polymer processing.

There are two fluxes of thermal energy that appear in the energy equation (the prefix "thermal" will be omitted, but implied from here on). One is molecular flux of energy and the other is convective flux of energy.

Molecular transport or Conduction

Molecular transport rates are adequately described for moderate temperature gradients by a linear relationship between the heat flux and the temperature gradient. The phenomenological relationship

$$\boxed{\boldsymbol{q} = -k\,\nabla T}$$

is known as Fourier's law. Here, $\boldsymbol{q}$ is the (vector) heat flux, and T is the temperature field at a given point. The negative sign tells us that heat flows in the direction opposite to that of the temperature gradient, namely from hot regions to cold regions. The constant of proportionality in Fourier's law, k is known as the thermal conductivity and is a material property of the fluid. The thermal conductivity depends on temperature and pressure in general, and Bird et al. provide some information regarding this subject in Chapter 9. Table B.2 provides results for the components of $\boldsymbol{q}$ in common coordinate systems.

Convective or Bulk Transport

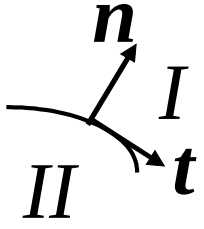
Thermal energy also is transported by the physical movement of an element of fluid from one place to another. This is known as convective transport. In our simplified picture, the (vector) convective flux of thermal energy at a given point can be written as

$$\text{Flux} = \rho C_p \boldsymbol{v}\left(T - T_{ref}\right)$$

where T_{ref} is a reference temperature that serves as a datum. A more precise accounting of the convective flux of total energy, is given in Bird et al. in Section 9.7.

Boundary Conditions

At the interface between a solid and a fluid, or that between a fluid and another fluid, it is reasonable to expect thermodynamic equilibrium to prevail between the two phases adjoining the interface, except when the heat flux across the interface is extremely large. A small portion of the interface between phases I and II is shown schematically in the sketch.



The assumption of thermodynamic equilibrium at the interface leads to the boundary condition

$$T_I = T_{II}$$

at the interface. In addition, because the interface is assumed to have no mass, any heat flux crossing the interface from one phase must necessarily be transmitted to the other phase. Therefore,

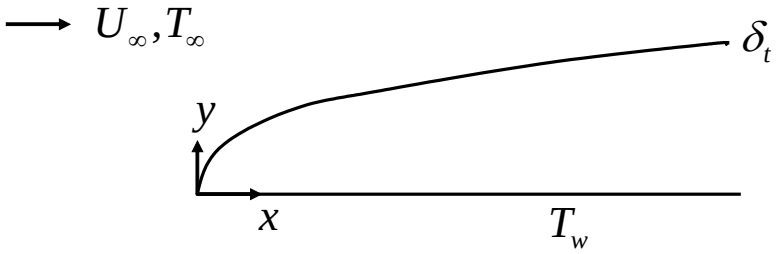
$$\boldsymbol{n} \bullet \boldsymbol{q}_I = \boldsymbol{n} \bullet \boldsymbol{q}_{II}$$

at the interface. The most general way to formulate a problem is to write the governing energy equation for each phase and the above pair of conditions at each phase interface, along with any other applicable initial and boundary conditions. In practice, it is far more convenient to study heat transport in controlled conditions wherein the temperature at a solid boundary in contact with a fluid is prescribed, or the heat flux from the solid to the fluid is prescribed. Experimentally, we can achieve a condition of prescribed uniform wall temperature in a pipe by choosing a highly conducting wall material and surrounding it with either a phase change system (such as condensing steam) or a segmented electrical heating system with a controller that maintains the temperature of the wall at a constant value. A uniform heat flux can be achieved most conveniently by using electrical heating. These two boundary conditions represent the two extremes of a family of boundary conditions that are common in heat transport problems.

The heat transfer coefficient

The heat transfer coefficient h and its dimensionless counterpart, the Nusselt Number Nu, are in common use in engineering work. Here, I discuss how the heat transfer coefficient is defined in typical situations.

Consider a fluid at a temperature T_∞ flowing at a uniform velocity U_∞ that encounters a rigid wall, which is maintained at a uniform temperature T_w. Let us assume that $T_w > T_\infty$ for the sake of definiteness; however, the results given below are equally valid when $T_w < T_\infty$.



Just as a momentum boundary layer forms at the wall and grows in thickness with distance x along the plate, a thermal boundary layer forms at the wall; the temperature of the fluid changes

from T_w to T_∞ in the thermal boundary layer. The thickness of the thermal boundary layer δ_t also grows with distance x.

At the rigid wall, the normal velocity is zero and the heat flux from the wall to the fluid consists only of the conduction flux q_y. From Fourier's law, this can be written as

$$q_w(x) = -k\frac{\partial T}{\partial y}(x,0)$$

Note that $q_w > 0$ when $T_w > T_\infty$, and $q_w < 0$ when $T_w < T_\infty$. Even though q_w is purely a conduction flux, it is modified by flow. As the velocity of the fluid increases, the ability of the fluid to carry away heat supplied by the plate increases, and the temperature gradient at the wall becomes sharper, consistent with a larger heat flux. You can see that a sharper temperature gradient at the wall is consistent with a thinner thermal boundary layer.

If we can solve the energy equation for the temperature distribution in the flowing fluid, the heat flux $q_w(x)$ can be evaluated as a function of x. A heat transfer coefficient $h(x)$ is defined for this system as follows.

$$q_w(x) = h(x)(T_w - T_\infty)$$

Therefore, the heat transfer coefficient is seen to be directly related to the temperature gradient at the wall.

$$h(x) = \frac{-k\frac{\partial T}{\partial y}(x,0)}{(T_w - T_\infty)}$$

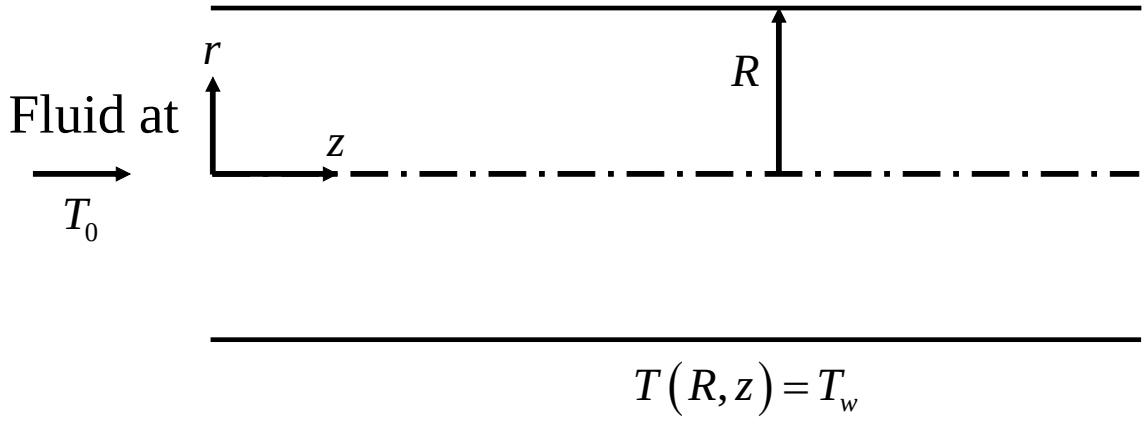
This definition of $h(x)$ holds regardless of the sign of $(T_w - T_\infty)$.

The Nusselt number Nu is a dimensionless version of the heat transfer coefficient. It is defined in the above problem as follows.

$$Nu = \frac{hL}{k}$$

Here L is a characteristic length scale, which can be taken as the axial distance x or the length of the plate, depending on our needs.

Now, consider heat transfer to a fluid flowing through a pipe. In this case, let the fluid enter the pipe at some temperature T_0, encountering a step change in wall temperature to T_w. A sketch of the system is given below.



At any given axial position z, the heat flux from the wall to the fluid is given by

$$q_w = k\frac{\partial T}{\partial r}(R,z)$$

Do you see why a positive sign is used in the right side? It is because the heat flux from the wall to the fluid is in the negative $r-$direction.

For defining the heat transfer coefficient, we need a driving force at any location z. While it is possible to use $(T_w - T_0)$, the more common choice is $(T_w - T_b)$ where T_b is known as the bulk or cup-mixing average temperature. The bulk average temperature is experimentally determined by collecting the fluid coming out of the system at a given axial location and mixing it completely, and then measuring its temperature. The following mathematical definition directly follows from this physical definition.

$$T_b = \frac{\int_0^R 2\pi r V(r) T(r,z)\,dr}{\int_0^R 2\pi r V(r)\,dr}$$

Here, $V(r)$ represents the velocity field.

The heat transfer coefficient in this system is defined as follows.

$$q_w = h(z)(T_w - T_b)$$

so that we can evaluate $h(z)$ using

$$h(z) = \frac{k\dfrac{\partial T}{\partial r}(R, z)}{(T_w - T_b)}$$

if we know the detailed temperature distribution in the fluid.

We see from these two examples that the heat transfer coefficient will depend on position, the system parameters, and on time in unsteady state problems. This concept of a heat transfer coefficient is extended to many practical heat transfer situations. Typically, in any given system, heat transfer rates and suitably defined driving forces are both measured. The ratio of the flux of thermal energy to the driving force expressed as a temperature difference is reported as the heat transfer coefficient.

$$h = \frac{\text{heat flux}}{\text{driving force}}$$

We can think of the heat transfer coefficient as the conductance (using an electrical analogy) of the system. Typically, spatial averages are easier to measure and report; it is not common in engineering design to use local values.

In problems amenable to analysis from first principles, temperature distributions and heat fluxes can be calculated directly from the solution. There is really no need to define a heat transfer coefficient. But, to make it convenient to report and use the results, even in such problems, suitably defined heat transfer coefficients are calculated from the theoretical results and reported.

Important dimensionless groups in heat transfer

We already have defined a dimensionless group commonly used in engineering practice, namely, the Nusselt number Nu. Considering it to be an average value for a given heat transfer setting such as heat transfer to fluid flowing through a circular pipe of diameter D, we can use dimensional analysis to identify the dimensionless groups on which Nu will depend. Thus, we obtain

$$Nu = Nu\left(\text{Re}, \text{Pr}, \frac{L}{D}\right)$$

where $Nu = \dfrac{hD}{k}$, the Reynolds number $\text{Re} = \dfrac{UD}{\nu}$, and the Prandtl number $\text{Pr} = \dfrac{\nu}{\alpha}$. The velocity U appearing in the Reynolds number is a characteristic velocity in the system such as the average velocity of flow, and ν and α are the kinematic viscosity and the thermal diffusivity of the fluid, respectively. The thermal diffusivity $\alpha = k/(\rho C_p)$. The symbol L stands for the length of the

pipe. For non-circular cross-sections, the Nusselt number will also depend on additional aspect ratio parameters.

The physical significance of the Nusselt number is simply that it represents a dimensionless heat flux at the wall or a ratio of the actual heat transfer rate to that prevailing in a hypothetical system in which the same driving force applied across the characteristic distance drives a conduction flux. We can see this by recasting it as follows.

$$Nu = \frac{h\Delta T}{k\dfrac{\Delta T}{D}}$$

The Prandtl number

To appreciate the physical significance of the Prandtl number, we note that it is the ratio of the intrinsic transport coefficients ν and α for molecular transport of momentum and energy, respectively. Consider a simple one-dimensional transport situation in which fluid flows in the $x-$direction with a velocity $v_x(y)$. In this case, Newton's law of viscosity for the momentum flux τ_{yx} (note that we are interpreting this symbol as the negative of the shear stress) can be written as

$$\tau_{yx} = -\mu\frac{\partial v_x}{\partial y}$$

and in a similar one-dimensional conduction problem, Fourier's law for the heat flux q_y can be written as

$$q_y = -k\frac{\partial T}{\partial y}$$

If we assume that the density and specific heat at constant pressure are constant, we can rewrite the above results in the following form.

$$\tau_{yx} = -\nu\frac{\partial(\rho v_x)}{\partial y}$$

$$q_y = -\alpha\frac{\partial\left(\rho C_p\left\{T - T_{ref}\right\}\right)}{\partial y}$$

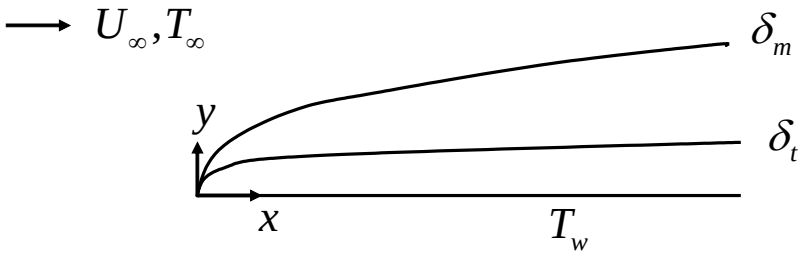
The product ρv_x represents the amount of $x-$momentum in unit volume of fluid, and can be regarded as the concentration of $x-$momentum. Therefore, we see that the flux of $x-$momentum is proportional to the gradient of the concentration of that momentum, and occurs in the direction opposite to that gradient. The coefficient of proportionality is the kinematic viscosity ν. In a like

manner, the flux of thermal energy is proportional to the gradient of the concentration of thermal energy, with a coefficient of proportionality equal to the thermal diffusivity α. So, molecular transport leads to momentum and energy flowing "downhill" in the direction opposite to that of the concentration gradient of each, with coefficients of proportionality that represent the ability of the fluid to transport momentum or energy by molecular means. This leads to the following physical interpretation of the Prandtl number.

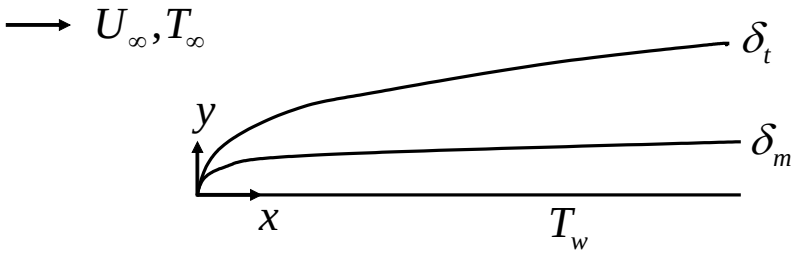
$$\mathrm{Pr} = \frac{\text{Ability of a fluid to transport momentum by molecular means}}{\text{Ability of that fluid to transport energy by molecular means}}$$

Thus, in flow over an object, the relative thicknesses of the momentum and thermal boundary layers reflect the magnitude of the Prandtl number, as the examples given below show.

Large Prandtl number, $\mathrm{Pr} \gg 1 \quad (\nu \gg \alpha)$



Small Prandtl number, $\mathrm{Pr} \ll 1 \quad (\nu \ll \alpha)$



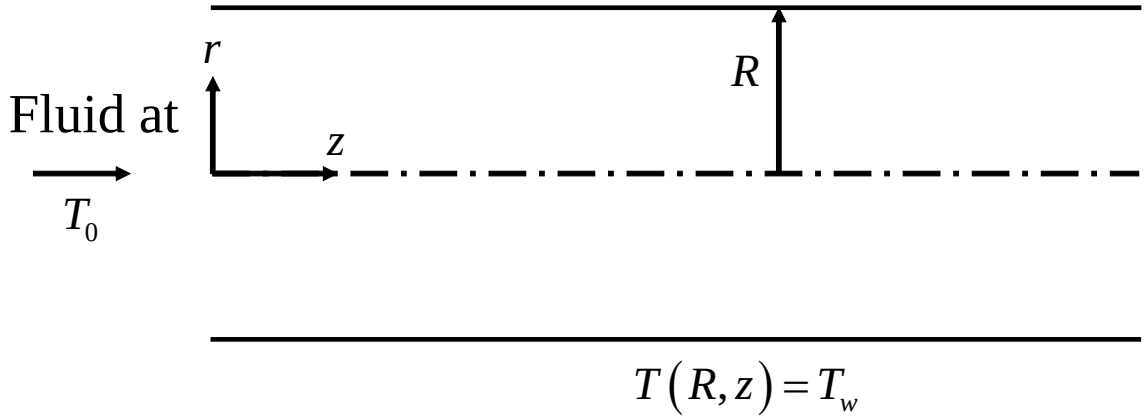
For gases, the Prandtl number is typically of $O(1)$, while for common liquids Prandtl numbers are found to vary from 10 to 1000. The Prandtl numbers of very viscous liquids such as polymer melts can be even larger than 1000, reaching values of 10^5 or greater. Because liquid metals are great conductors of thermal energy, Prandtl numbers for liquid metals are typically of $O(10^{-2})$.

Part VII

Graetz Problem
Lévêque Approximation

The Graetz Problem

As a good model problem, we consider steady state heat transfer to fluid in steady flow through a tube. The fluid enters the tube at a temperature T_0 and encounters a wall temperature at T_w, which can be larger or smaller than T_0. A simple version of this problem was first analyzed by Graetz (1883). A sketch of the system is shown below.



Objective

To obtain the steady temperature distribution $T(r,z)$ in the fluid, and to calculate the rate of heat transfer from the wall to the fluid

Assumptions

1. Steady fully developed laminar flow; steady temperature field.

2. Constant physical properties ρ, μ, k, C_p -- This assumption also implies incompressible Newtonian flow.

3. Axisymmetric temperature field $\Rightarrow \dfrac{\partial T}{\partial \varphi} \equiv 0$, where we are using the symbol φ for the polar angle. This is because we want to use the symbol θ to represent dimensionless temperature later.

4. Negligible viscous dissipation

Velocity Field

Poiseuille Flow

$v_r = 0; \qquad v_\varphi = 0$

$v_z(r) = v_0\left(1-\frac{r^2}{R^2}\right)$ $\qquad v_0$: Maximum velocity existing at the centerline

Energy Equation

Subject to assumption (2), Equation (B.9.2) from Bird et al. (page 850) can be written as follows.

$$\rho C_p\left[\cancel{\frac{\partial T}{\partial t}} + \cancel{v_r}\frac{\partial T}{\partial r} + \frac{\cancel{v_\varphi}}{r}\frac{\partial T}{\partial \varphi} + v_z\frac{\partial T}{\partial z}\right] = k\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right) + \frac{1}{r^2}\cancel{\frac{\partial^2 T}{\partial \varphi^2}} + \frac{\partial^2 T}{\partial z^2}\right] + \cancel{\mu\Phi_v}$$

(1) $v_r = 0$ $v_\varphi = 0$ (3) (4)

and therefore, simplified to

$$v_0\left(1-\frac{r^2}{R^2}\right)\frac{\partial T}{\partial z} = \alpha\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right) + \frac{\partial^2 T}{\partial z^2}\right]$$

where $\alpha = k/(\rho C_p)$ is the thermal diffusivity of the fluid.

Boundary Conditions

Inlet: $\qquad T(r,0) = T_0$

Wall: $\qquad T(R,z) = T_w$

Centerline: $\qquad T(0,z)$ is finite

or $\qquad \frac{\partial T}{\partial r}(0,z) = 0$

Because of the appearance of the axial conduction term in the governing differential equation, we should write another boundary condition in the $z-$coordinate. But actually, the inlet condition

written above is incompatible with the inclusion of axial conduction in the problem, because conduction will lead to some of the information about the step change in wall temperature at the inlet to propagate backward. As we shall see shortly, we'll neglect axial conduction, which will obviate the need for writing a second condition in the $z-$coordinate.

Non-Dimensionalization

We shall use the following scheme for scaling (or non-dimensionalizing) the variables.

$$\theta = \frac{T - T_w}{T_0 - T_w}\ , \quad Y = \frac{r}{R}, \quad Z = \frac{z}{R\,Pe}, \quad \text{where the Péclet Number } Pe = \frac{Rv_0}{\alpha}.$$

This permits us to transform the governing differential equation and boundary conditions to the following form.

$$\left(1 - Y^2\right)\frac{\partial \theta}{\partial Z} = \frac{1}{Y}\frac{\partial}{\partial Y}\left(Y\frac{\partial \theta}{\partial Y}\right) + \frac{1}{Pe^2}\frac{\partial^2 \theta}{\partial Z^2}$$

$$\theta(Y,0) = 1$$
$$\theta(1,Z) = 0$$
$$\theta(0,Z) \text{ is finite}$$

or $$\frac{\partial \theta}{\partial Y}(0,Z) = 0$$

The Péclet Number

The Péclet number plays the same role in heat transport as the Reynolds number does in fluid mechanics. First, we note that the Péclet number is the product of the Reynolds and Prandtl numbers.

$$Pe = \frac{Rv_0}{\alpha} = \frac{Rv_0}{\nu} \times \frac{\nu}{\alpha} = \text{Re} \times \text{Pr}$$

The physical significance of the Péclet number can be inferred by recasting it slightly.

$$Pe = \frac{\rho v_0 C_p \Delta T}{k\dfrac{\Delta T}{R}} = \frac{\text{Rate of energy transport by convection}}{\text{Rate of energy transport by conduction}}$$

Note that the numerator represents the order of magnitude of the convective flux in the main flow direction, whereas the denominator stands for the order of magnitude of the conduction flux in the radial direction. If we wish to compare the rates of energy transport by these two mechanisms in the same direction, we can multiply the Péclet number by L/R where L is a characteristic length in the axial direction.

For large values of Pe, we can see that $\frac{1}{Pe^2} \ll 1$. Therefore, in the scaled energy equation, the term involving axial conduction can be safely neglected. Physically, there are two mechanisms for transporting energy in the axial direction, namely, convection and conduction. Because the Péclet number is large, we are able to neglect transport by conduction in comparison with transport by convection. On the other hand, in the radial direction, there is only a single mechanism for transport of energy, namely conduction. By performing calculations including conduction in the axial direction, it has been established that it is safe to neglect axial conduction for $Pe \geq 100$. To learn about how to include axial conduction, you can consult the articles by Davis (1973), Acrivos (1980), and Papoutsakis et al. (1980).

Let us make a sample calculation of the Péclet number for laminar flow heat transfer in a tube. The thermal diffusivity of common liquids is typically in the range $10^{-7} - 2\times10^{-7}\ \frac{m^2}{s}$, and we'll use the larger limit. Choose

$$R = 10\ mm, \quad v_0 = 0.05\ \frac{m}{s}, \quad \alpha = 2\times10^{-7}\ \frac{m^2}{s}$$

This yields, $Pe = 2,500$, which is much larger than 100. We can check to see if the flow is laminar by calculating the Reynolds number. If the fluid is water, $\nu \approx 10^{-6}\ \frac{m^2}{s}$, which yields a Prandtl number $\mathrm{Pr} = \frac{\nu}{\alpha} = 5$. Therefore, the Reynolds number is $\mathrm{Re} = 500$, which is comfortably in the laminar flow regime.

The final version of the scaled energy equation is

$$\left(1-Y^2\right)\frac{\partial\theta}{\partial Z} = \frac{1}{Y}\frac{\partial}{\partial Y}\left(Y\frac{\partial\theta}{\partial Y}\right)$$

We can solve this equation by separation of variables, because the boundary conditions in the Y – coordinate are homogeneous. The method of separation of variables yields an infinite series solution for the scaled temperature field.

$$\theta(Y,Z) = \sum_{n=1}^{\infty} A_n e^{-\lambda_n^2 Z}\phi_n(Y)$$

In the above solution, the functions $\phi_n(Y)$ are the characteristic functions or eigenfunctions of a proper Sturm-Liouville system.

$$\frac{1}{Y}\frac{d}{dY}\left(Y\frac{d\phi}{dY}\right) + \lambda^2\left(1-Y^2\right)\phi = 0$$

$\frac{d\phi}{dY}(0)=0$ or $\phi(0)$ is finite

$\phi(1)=0$

The above ordinary differential equation for $\phi(Y)$ can be solved by applying the following transformations to both the dependent and the independent variables (Lauwerier, 1951, Davis, 1973).

$X=\lambda Y^2 \qquad W(X)=e^{\frac{X}{2}}\phi(Y)$

This leads to the following differential equation for $W(X)$.

$$X\frac{d^2W}{dX^2}+(1-X)\frac{dW}{dX}+\left(\frac{\lambda}{4}-\frac{1}{2}\right)W=0$$

This is known as Kummer's equation. It has two linearly independent solutions, but only one is bounded at $X=0$. Because $\phi(0)$ must be bounded, we must require that $W(0)$ also remain bounded. This rules out the singular solution, leaving us with the regular solution

$$W(X)=c\,M\left(\frac{1}{2}-\frac{\lambda}{4},\ 1,\ X\right)$$

where c is an arbitrary multiplicative constant. The function $M(a,b,X)$ is the confluent hypergeometric function, or Kummer function, and is discussed in Chapter 13 of the "Handbook of Mathematical Functions" by M. Abramowitz and I. A. Stegun, It is an extension of the exponential function, and is written in the form of the following series.

$$M(a,b,X)=1+\frac{a}{b}X+\frac{a(a+1)}{b(b+1)}\frac{X^2}{2!}+\cdots$$
$$+\frac{a(a+1)\cdots(a+n-1)}{b(b+1)\cdots(b+n-1)}\frac{X^n}{n!}+\cdots$$

You can see that when $a=b$,

$$M(a,a,X)=e^X$$

Application of the boundary condition at the tube wall, $\phi(1)=0$, leads to the following transcendental equation for the eigenvalues.

$$M\left(\frac{1}{2}-\frac{\lambda}{4},\ 1,\ \lambda\right)=0$$

The above equation has infinitely many discrete solutions for λ, which we designate as λ_n, with n assuming positive integer values beginning from 1. Corresponding to each value λ_n, there is an eigenfunction $\phi_n(Y)$ given by

$$\phi_n(Y) = e^{-\frac{\lambda_n Y^2}{2}} W_n\left(\lambda_n Y^2\right)$$

The first few eigenvalues are reported in the table.

n	λ_n	λ_n^2
1	2.7044	7.3136
2	6.6790	44.609
3	10.673	113.92
4	14.671	215.24
5	18.670	348.57

Note that technically $\left\{\lambda_n^2\right\}$ is the set of eigenvalues, even though we use the term loosely to designate $\left\{\lambda_n\right\}$ as that set for convenience.

The most important property of a proper Sturm-Liouville system is that the eigenfunctions are orthogonal with respect to a weighting function that is specific to that system. In the present case, the orthogonality property of the eigenfunctions can be stated as follows.

$$\int_0^1 \phi_m(Y)\phi_n(Y)Y\left(1-Y^2\right)dY = 0, \quad m \neq n$$

Using this orthogonality property, it is possible to obtain a result for the coefficients in the solution by separation variables.

$$A_n = \frac{\int_0^1 \phi_n(Y)Y\left(1-Y^2\right)dY}{\int_0^1 \phi_n^2(Y)Y\left(1-Y^2\right)dY}$$

The Heat Transfer Coefficient

The heat flux from the wall to the fluid, $q_w(z)$ is a function of axial position. It can be calculated directly by using the result

$$q_w(z) = k\frac{\partial T}{\partial r}(R, z)$$

but as we noted earlier, it is customary to define a heat transfer coefficient $h(z)$ via

$$q_w(z) = h(z)\left(T_w - T_b\right)$$

where the bulk or cup-mixing average temperature T_b is introduced. The way to experimentally determine the bulk average temperature is to collect the fluid coming out of the system at a given axial location, mix it completely, and measure its temperature. The mathematical definition of the bulk average temperature was given in an earlier section.

$$T_b = \frac{\int_0^R 2\pi r V(r) T(r,z)\, dr}{\int_0^R 2\pi r V(r)\, dr}$$

where the velocity field $V(r) = v_0\left(1 - r^2/R^2\right)$. You can see from the definition of the heat transfer coefficient that it is related to the temperature gradient at the tube wall in a simple manner.

$$h(z) = \frac{k\dfrac{\partial T}{\partial r}(R,z)}{\left(T_w - T_b\right)}$$

We can define a dimensionless heat transfer coefficient, which is known as the Nusselt number.

$$Nu(Z) = \frac{2hR}{k} = -2\,\frac{\dfrac{\partial \theta}{\partial Y}(1,Z)}{\theta_b(Z)}$$

where θ_b is the dimensionless bulk average temperature.

By substituting from the infinite series solution for both the numerator and the denominator, the Nusselt number can be written as follows.

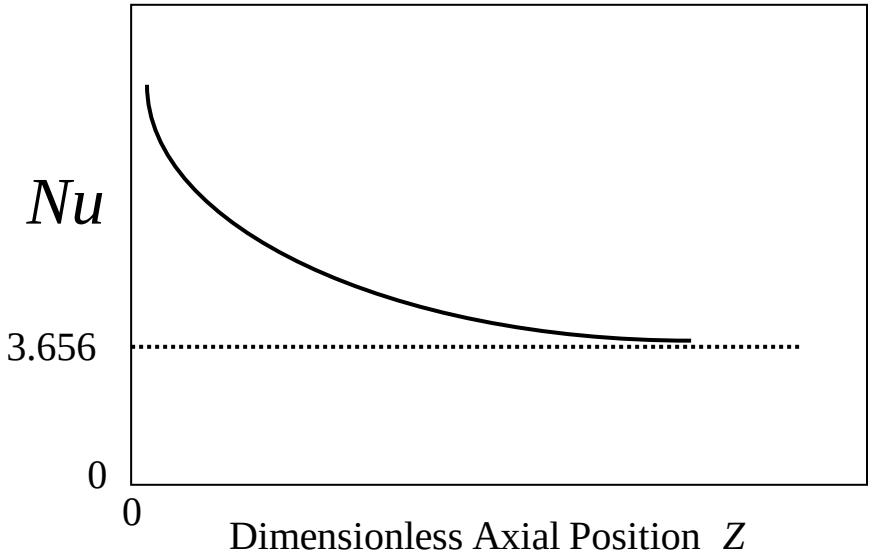
$$Nu(Z) = -2\,\frac{\displaystyle\sum_{n=1}^{\infty} A_n e^{-\lambda_n^2 Z}\,\frac{d\phi_n}{dY}(1)}{4\displaystyle\sum_{n=1}^{\infty} A_n e^{-\lambda_n^2 Z}\int_0^1 Y\left(1-Y^2\right)\phi_n(Y)\,dY}$$

The denominator can be simplified by using the governing differential equation for $\phi_n(Y)$, along with the boundary conditions, to finally yield the following result.

$$Nu = \frac{\displaystyle\sum_{n=1}^{\infty} A_n e^{-\lambda_n^2 Z}\,\frac{d\phi_n}{dY}(1)}{2\displaystyle\sum_{n=1}^{\infty} A_n \frac{e^{-\lambda_n^2 Z}}{\lambda_n^2}\,\frac{d\phi_n}{dY}(1)}$$

We can see that for large Z, only the first term in the infinite series in the numerator, and likewise the first term in the infinite series in the denominator, is important. Therefore, as $Z \to \infty$, $Nu \to \frac{\lambda_1^2}{2} = 3.656$.

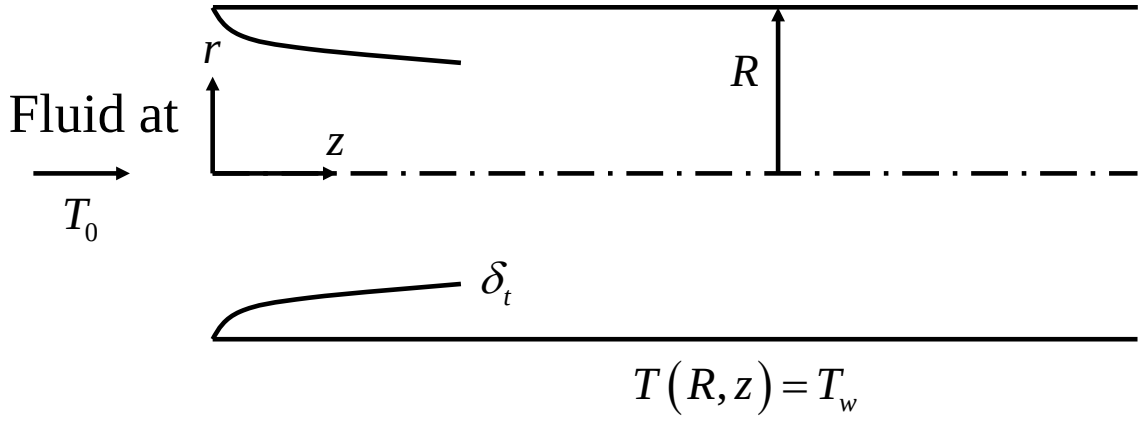
The sketch qualitatively illustrates the behavior of the Nusselt number as a function of dimensionless axial position.



A similar analysis is possible in the case of a uniform wall flux boundary condition. Extensions of the Graetz solution by separation of variables have been made in a variety of ways, accommodating non-Newtonian flow, turbulent flow, and other geometries besides a circular tube.

The Lévêque Approximation

The orthogonal function expansion solution obtained above is convergent at all values of the axial position, but convergence is very slow as the inlet is approached. The main reason for this is the assistance provided by $e^{-\lambda_n^2 Z}$ in accelerating convergence for sufficiently large values of Z. Lévêque (1928) considered the thermal entrance region in a tube and developed an alternative solution, which is useful precisely where the orthogonal function expansion converges too slowly.



We shall now construct the Lévêque solution which is built on the assumption that the thickness of the thermal boundary layer $\delta_t \ll R$. This assumption leads to the following simplifications.

1. Curvature effects can be neglected in the radial conduction term. This means that the derivative $\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right)$ can be approximated by $\frac{1}{R}\frac{\partial}{\partial r}\left(R\frac{\partial T}{\partial r}\right)=\frac{\partial^2 T}{\partial r^2}$.

2. Because we are only interested in the velocity distribution within the thermal boundary layer, we expand the velocity field in a Taylor series in distance measured from the tube wall and retain the first non-zero term.

Defining $x = R - r$, we can rewrite the velocity distribution as

$$v_z(r) = v_0\left(1-\frac{(R-x)^2}{R^2}\right)=v_0\left(2\frac{x}{R}-\frac{x^2}{R^2}\right)\approx 2v_0\frac{x}{R}$$

Recall that a power series obtained by any method is a Taylor series. The above approach is simpler than working out the derivatives of $v_z(r)$ in the $x-$coordinate, evaluating them at the wall, and constructing the Taylor series.

3. Because the conditions outside the thermal boundary layer are those in the fluid entering the tube, we shall use the boundary condition $T(x \to \infty) \to T_0$ instead of the centerline boundary condition employed in obtaining the Graetz solution.

Beginning with the simplified energy equation in which axial conduction has been neglected already, and invoking the above assumptions, we have the following governing equation for the temperature field.

$$2v_0 \frac{x}{R} \frac{\partial T}{\partial z} = \alpha \frac{\partial^2 T}{\partial x^2}$$

where the chain rule has been used to transform the second derivative in r to the second derivative in x.

The temperature field $T(x,z)$ satisfies the following boundary conditions.

$$T(x,0) = T_0$$
$$T(0,z) = T_w$$
$$T(\infty,z) = T_0$$

We shall work with a dimensionless version of these equations. For consistency, we scale the temperature and axial coordinate in the same manner as before.

$$\theta = \frac{T - T_w}{T_0 - T_w} \qquad Z = \frac{z}{R\,Pe}$$

We define a new scaled distance from the wall via $X = x/R$. The scaled governing equation and boundary conditions are given below.

$$2X \frac{\partial \theta}{\partial Z} = \frac{\partial^2 \theta}{\partial X^2}$$

$$\theta(X,0) = 1$$
$$\theta(0,Z) = 0$$
$$\theta(\infty,Z) = 1$$

The similarity of this governing equation and boundary conditions to those in the fluid mechanical problem in which we solved for the velocity distribution between two plates when one of them is held fixed and the other is moved suddenly is not a coincidence. For small values of time in the fluid mechanical problem, we replaced the boundary condition at the top plate with one at an infinite distance from the suddenly moved plate, and used the method of combination of variables to solve the equations. It would be worthwhile for you to go back and review the notes on "combination of variables" at this stage.

By invoking ideas very similar to those used in the fluid mechanical problem, we postulate that a similarity solution exists for the temperature field in the present problem. That is, we assume $\theta(X,Z)=F(\eta)$ where the similarity variable $\eta = X/\delta(Z)$. The variable $\delta(Z)$ represents the scaled thermal boundary layer thickness, and is unknown at this stage. We make the necessary transformations using the chain rule.

$$\frac{\partial\theta}{\partial Z} = \frac{\partial\eta}{\partial Z}\frac{dF}{d\eta} = \left(-\frac{X}{\delta^2}\frac{d\delta}{dZ}\right)\frac{dF}{d\eta} = -\frac{\eta}{\delta}\frac{d\delta}{dZ}\frac{dF}{d\eta}$$

$$\frac{\partial\theta}{\partial X} = \frac{\partial\eta}{\partial X}\frac{dF}{d\eta} = \frac{1}{\delta}\frac{dF}{d\eta}$$

$$\frac{\partial^2\theta}{\partial X^2} = \frac{\partial}{\partial X}\left[\frac{1}{\delta(Z)}\frac{dF}{d\eta}\right] = \frac{1}{\delta}\frac{\partial}{\partial X}\left[\frac{dF}{d\eta}\right] = \frac{1}{\delta}\frac{\partial\eta}{\partial X}\frac{d}{d\eta}\left[\frac{dF}{d\eta}\right] = \frac{1}{\delta^2}\frac{d^2F}{d\eta^2}$$

Using these results, the partial differential equation for $\theta(X,Z)$ is transformed to an ordinary differential equation for $F(\eta)$.

$$\frac{d^2F}{d\eta^2} + 2\eta^2\left(\delta^2\frac{d\delta}{dZ}\right)\frac{dF}{d\eta} = 0$$

It is evident that the similarity hypothesis will fail unless the quantity inside the parentheses is required to be independent of Z, and therefore, a constant. For convenience, we set this constant to 3/2. Therefore, we have an ordinary differential equation for $F(\eta)$ and another for $\delta(Z)$.

$$\frac{d^2F}{d\eta^2} + 3\eta^2\frac{dF}{d\eta} = 0$$

$$\delta^2\frac{d\delta}{dZ} = \frac{3}{2}$$

To derive the boundary conditions on these functions, we must go to the boundary conditions on $\theta(X,Z)$. In a straightforward way, we see that $\theta(0,Z)=0$ yields $F(0)=0$, and $\theta(\infty,Z)=1$ leads to $F(\infty)=1$. The remaining (inlet) condition gives

$$\theta(X,0) = F\left(\frac{X}{\delta(0)}\right) = 1$$

By choosing $\delta(0)=0$, this condition collapses into the condition $F(\infty)=1$ obtained already from the boundary condition on the scaled temperature field as $X\to\infty$. Summarizing the boundary conditions on $F(\eta)$ and $\delta(Z)$, we have

$F(0)=0,\quad F(\infty)=1,$ and

$\delta(0)=0$

Integration yields the following solution for the scaled boundary layer thickness $\delta(Z)$.

$$\delta(Z)=\left(\frac{9}{2}Z\right)^{1/3}$$

The solution for $F(\eta)$ is

$$F(\eta)=\frac{\int_0^{\eta}e^{-\gamma^3}d\gamma}{\int_0^{\infty}e^{-\gamma^3}d\gamma}=\frac{1}{\Gamma(4/3)}\int_0^{\eta}e^{-\gamma^3}d\gamma$$

Here, $\Gamma(x)$ represents the Gamma function, discussed in the "Handbook of Mathematical Functions" by Abramowitz and Stegun. The numerical value of $\Gamma(4/3)\approx 0.89298$, so that we can write $1/\Gamma(4/3)\approx 1.1199$ or roughly 1.120.

Heat Transfer Coefficient

In the thermal entrance region, when the thermal boundary layer is thin, we can approximate the bulk average temperature T_b by the temperature of the fluid entering the tube T_0. Therefore, we define the heat transfer coefficient in this entrance region by

$$q_w=k\frac{\partial T}{\partial r}(R,z)=h(T_w-T_0)$$

Transforming to dimensionless variables, and defining a Nusselt number $Nu=2hR/k$, we can write

$$Nu(Z)=2\frac{\partial\theta}{\partial X}(0,Z)=\frac{2}{\delta(Z)}\frac{dF}{d\eta}(0)$$

By substituting for $\delta(Z)$ and $\frac{dF}{d\eta}(0)$, we obtain the following approximate result for the Nusselt number in the thermal entrance region.

$$Nu(Z) \approx 1.357\, Pe^{1/3} \left(\frac{R}{z}\right)^{1/3}$$

Comparison with the exact solution shows this result is a good approximation in the range

$$\frac{Pe}{2500} \leq \left(\frac{z}{R}\right) \leq \frac{Pe}{50}$$

References

M. Abramowitz and I. Stegun, Handbook of Mathematical Functions, Dover, 1965.

A. Acrivos, The extended Graetz problem at low Péclet numbers, Appl. Sci. Res. **36**, 35 (1980).

E.J. Davis, Can. J. Chem. Engg. **51**, 562 (1973).

L. Graetz, Ueber die Wärmeleitungsfähigkeit von Flüssigkeiten, Annalen der Physik **254**, 79 (1882). DOI: 10.1002/andp.18822540106

H.A. Lauwerier, Appl. Sci. Res. **A2**, 184 (1951).

M.A. Lévêque, Les lois de la transmission de chaleur par convection, Annales des Mines, Memoires, Series 12, **13**, 201-299, 305-362, 381-415 (1928) {as cited by J. Newman, Trans. ASME J. Heat Transfer, 91, 177 (1969)}

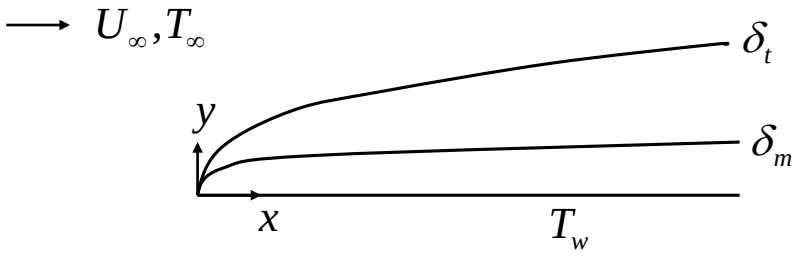
E. Papoutsakis, D. Ramkrishna, and H.C. Lim, The extended Graetz problem with Dirichlet wall boundary conditions, Appl. Sci. Res. **36**, 13 (1980).

Part VIII

Thermal Boundary Layer
Buoyant Convection
Surface-Tension-Driven Flows

Thermal boundary layer over a flat plate

The problem of laminar flow heat transfer between a flowing fluid and a flat plate that is wide and long is analyzed. Suitable assumptions are made so that two-dimensional problems can be posed for both the fluid mechanical and heat transfer situations. A sketch of the system is given below.



As seen from the sketch, the fluid approaches the plate with a velocity U_∞ and temperature T_∞. The surface of the plate is maintained at a uniform temperature T_w. Boundary layers form on the surface. They represent regions in which the velocity and temperature change from the values at the surface of the plate to those in the free stream. The relative thickness of the thermal boundary layer, $\delta_t(x)$, when compared with the thickness of the momentum boundary layer $\delta_m(x)$, will depend on the magnitude of the Prandtl number $\mathrm{Pr} = \nu/\alpha$, where ν is the kinematic viscosity and α represents the thermal diffusivity of the fluid. In the sketch shown, the Prandtl number is small compared with unity, so that the thermal boundary layer grows more rapidly with distance along the plate than the momentum boundary layer.

A list of simplifying assumptions follows next.

Assumptions

1. Steady velocity and temperature fields

2. Constant density ρ, viscosity μ, thermal conductivity k, and specific heat C_p

3. The system is wide in the $z-$direction, permitting edge effects to be neglected. This implies that $\dfrac{\partial \mathbf{v}}{\partial z} \equiv \mathbf{0}$; $\dfrac{\partial T}{\partial z} \equiv 0$; $w \equiv 0$.

4. Conduction in the $x-$direction is negligible compared with convective transport in the same direction.

5. Viscous dissipation is negligible, and there are no sources or sinks.

6. All the assumptions of Prandtl's boundary layer theory are applicable to the momentum transport problem. It is assumed that the solution of that problem is available.

Energy equation

The simplification of the energy equation in rectangular Cartesian coordinates is shown below.

$$\rho C_p\left(\cancel{\frac{\partial T}{\partial t}}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+v_z\cancel{\frac{\partial T}{\partial z}}\right)=k\left(\cancel{\frac{\partial^2 T}{\partial x^2}}+\frac{\partial^2 T}{\partial y^2}+\cancel{\frac{\partial^2 T}{\partial z^2}}\right)+\mu\,\cancel{\Phi_v}+\cancel{S}$$

(1) (3) (4) (3) (5) (5)

Therefore, the energy equation reduces to

$$\boxed{u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha\frac{\partial^2 T}{\partial y^2}}$$

The boundary conditions on the temperature field are listed below.

$T(0,y)=T_\infty$ Temperature of entering fluid is specified

$T(x,0)=T_w$ Prescribed temperature at the surface of the plate

$T(x,\infty)=T_\infty$ Temperature far from the wall is that of entering fluid

The governing equation for a dimensionless temperature $\theta=\dfrac{T-T_w}{T_\infty-T_w}$ is

$$\boxed{u\frac{\partial \theta}{\partial x}+v\frac{\partial \theta}{\partial y}=\alpha\frac{\partial^2 \theta}{\partial y^2}}$$

and the boundary conditions become

$\theta(0,y)=1$

$\theta(x,0)=0$

$\theta(x,\infty)=1$

Now, let us examine the governing equation and boundary conditions for a dimensionless velocity $U=u(x,y)/U_\infty$.

$$\boxed{u\frac{\partial U}{\partial x}+v\frac{\partial U}{\partial y}=\nu\frac{\partial^2 U}{\partial y^2}}$$

$U(0,y)=1$

$U(x,0)=0$

$U(x,\infty)=1$

It is evident that the equations for $U(x,y)$ are identical to those for $\theta(x,y)$, except for the appearance of the kinematic viscosity ν in the momentum equation in place of the thermal diffusivity α multiplying the molecular transport term in the energy equation. If the Prandtl number were to be unity, the equations would indeed be identical.

For an arbitrary value of the Prandtl number, the method of combination of variables that was employed in the momentum boundary layer problem can be employed for the solution of the energy equation. If we let $\theta(x,y) = H(\eta)$ where the similarity coordinate $\eta = y/\sqrt{\nu x/U_\infty}$, the chain rule can be used to transform the energy equation to

$$\frac{d^2H}{d\eta^2} + \frac{\mathrm{Pr}}{2} f(\eta) \frac{dH}{d\eta} = 0$$

along with $H(0) = 0$ and $H(\infty) = 1$, Here the function $f(\eta)$ comes from the solution of the momentum boundary layer problem. If we let $U(x,y) = g(\eta)$, then $f(\eta) = \int_0^\eta g(\gamma)\, d\gamma$.

The function $f(\eta)$ satisfies

$$\frac{d^2 f}{d\eta^2} + \frac{1}{2} f \frac{df}{d\eta} = 0$$

along with $f(0) = 0;\ f'(0) = 0;\ f'(\infty) = 1$. This solution is obtained numerically, and is available in textbooks on fluid mechanics.

The solution for $H(\eta)$ is given below.

$$H(\eta) = \frac{\int_0^\eta \exp\left(-\frac{\mathrm{Pr}}{2} \int_0^\lambda f(\gamma) d\gamma \right) d\lambda}{\int_0^\infty \exp\left(-\frac{\mathrm{Pr}}{2} \int_0^\lambda f(\gamma) d\gamma \right) d\lambda}$$

Nusselt number

We can define a heat transfer coefficient h_x using the result for the local heat flux from the plate surface to the fluid.

$$q_w = -k \frac{\partial T}{\partial y}(x,0) = h_x (T_w - T_\infty)$$

The Nusselt number $Nu_x = h_x x/k$ is given by

$$Nu_x = -\frac{x\dfrac{\partial T}{\partial y}(x,0)}{(T_w - T_\infty)} = \sqrt{\frac{xU_\infty}{\nu}}\,\frac{dH}{d\eta}(0) = \mathrm{Re}_x^{1/2}\,H'(0)$$ where the Reynolds number $\mathrm{Re}_x = \dfrac{xU_\infty}{\nu}$. Therefore, we see that the heat transfer coefficient $h_x \propto \dfrac{1}{\sqrt{x}}$ for a laminar thermal boundary layer.

From Kays and Crawford (1980), over a good range of moderate values of the Prandtl number $(0.5 \le \mathrm{Pr} \le 15)$, the result for $H'(0)$ can be fitted to yield

$$\boxed{Nu_x = 0.332\,\mathrm{Re}_x^{1/2}\,\mathrm{Pr}^{1/3}}$$

Special cases

1. $\mathrm{Pr} \gg 1$

When the Prandtl number is very large, i.e., $\mathrm{Pr} \gg 1$, the thermal boundary layer is very thin compared with the momentum boundary layer. Therefore, the velocity distribution can be adequately approximated by a straight line within the thermal boundary layer. In this case, analytical solution of the energy equation is possible, and yields $\theta(\eta) = \dfrac{f''(0)}{2}\eta^2$. From the momentum boundary layer equations, $f''(0) = 0.3321$. As a result, we obtain

$$\boxed{Nu_x \approx 0.339\,\mathrm{Re}_x^{1/2}\,\mathrm{Pr}^{1/3}}$$

2. $\mathrm{Pr} \ll 1$

In the case of liquid metals, which are excellent thermal conductors, the Prandtl number is typically of $O(10^{-2})$, and therefore satisfies the requirement $\mathrm{Pr} \ll 1$. In this situation, the thermal boundary layer is much thicker than the momentum boundary layer. As a result, the velocity distribution is uniform with $u = U_\infty$ over most of the thermal boundary layer, with a deviation only in a thin region close to the flat plate. By approximating u by U_∞ everywhere in the thermal boundary layer, an analytical solution can be obtained for the scaled temperature field. From this, the following result can be obtained for the Nusselt number

$$\boxed{Nu_x \approx 0.565\,\mathrm{Re}_x^{1/2}\,\mathrm{Pr}^{1/2} = 0.565\,\mathrm{Pe}_x^{1/2}}$$

where $\mathrm{Pe}_x = xU_\infty/\alpha$ is the Peclet number for heat transfer.

Reference

W.M. Kays and M.E. Crawford, Convective Heat and Mass Transfer, McGraw-Hill, 1980.

The Boussinesq Approximation

When a fluid is heated, its density generally decreases. When density variations occur in a fluid, and the temperature differences that cause these variations are generally small, the resulting variations in density are small compared with the density itself. This permits one to make an important approximation in solving the problem. This approximation, attributed to Boussinesq, is stated as follows. The density of the fluid ρ is treated as a constant everywhere in the governing equations with one exception. That exception is made in the body force term in the Navier-Stokes equation. Thus, the use of the Boussinesq approximation, in conjunction with the assumption that the physical properties are otherwise constant, leads to the following set of governing equations.

Continuity

$$\nabla \bullet \boldsymbol{v} = 0$$

Navier-Stokes

$$\rho_0 \frac{d\boldsymbol{v}}{dt} = -\nabla p + \rho(\boldsymbol{x}) g + \mu \nabla^2 \boldsymbol{v}$$

Energy

$$\rho_0 C_p \frac{dT}{dt} = k \nabla^2 T + \mu \Phi_v + S$$

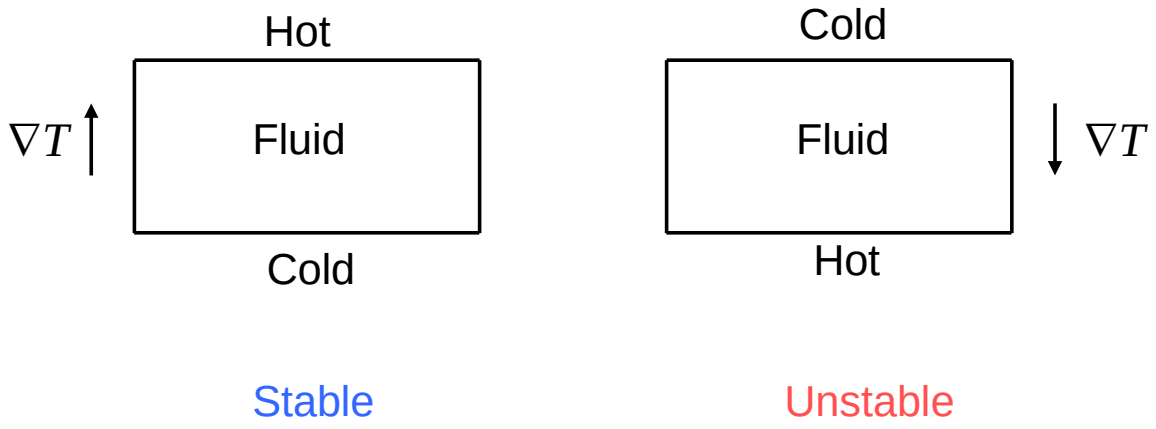
Reference

S. Chandrasekhar, Hydrodynamic and Hydromagnetic Stability, Oxford University Press, 1961 (reprinted in 1981 by Dover).

Buoyant Convection

"Natural" or "Buoyant" or "Free" convection is a very important mechanism that is operative in a variety of environments from cooling electronic circuit boards in computers to causing large scale circulation in the atmosphere as well as in lakes and oceans that influences the weather. It is caused by the action of density gradients in conjunction with a gravitational field. This is a brief introduction that will help you understand the qualitative features of a variety of situations you might encounter. A good reference book for natural convection flows is that by Gebhart et al.

There are two basic scenarios in the context of natural convection. In one, a density gradient exists in a fluid in a direction that is parallel to the gravity vector or opposite to it. Such situations can lead to "stable" or "unstable" density stratification of the fluid. In a stable stratification, less dense fluid is at the top and more dense fluid at the bottom. In the absence of other effects, convection will be absent, and we can treat the heat transfer problem as one of conduction. In an unstable stratification, in which less dense fluid is at the bottom, and more dense fluid at the top, provided the density gradient is sufficiently large, cellular convection will start spontaneously.



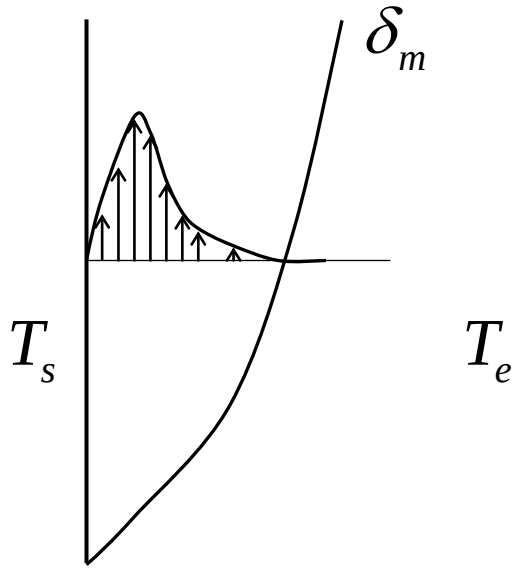
You should note that density gradients can arise not only from temperature gradients, but also from composition gradients even in an isothermal system. Here, we restrict our discussion to the case when temperature gradients are the source of the density gradients.

The more common situation that we encounter in heat transfer is one in which there is a density gradient perpendicular to the gravity vector. Consider a burning candle. The air next to the hot candle flame is hot, whereas the air laterally farther from it is relatively cooler. This will set up a natural convection flow around the candle, in which the cool surrounding air approaches the surface of the candle, rises, and flows in a hot plume above the flame. It is this flow that causes the visible flame to take the shape it does; by the way, the flame is simply gas at such a high temperature that it radiates visible light. In the absence of gravity, a candle flame would be spherical.

Another example is the flow of air at the tip of a lit cigarette; in this case, the smoke from the cigarette actually traces that flow for us. In a common technique used for home heating, the baseboard heater consists of a tube through which hot water flows, and the heater is placed close to the floor. The tube is outfitted with fins to provide additional heat transfer surface. The neighboring air is heated, and the hot air rises, with cooler air moving in toward the baseboard at floor level. This natural convection circulation set up by the hot baseboard provides a simple mixing mechanism for the air in the room and helps us maintain a relatively uniform temperature everywhere. Clearly, the convection helps the heat transfer process here.

Natural Convection adjacent to a heated vertical surface

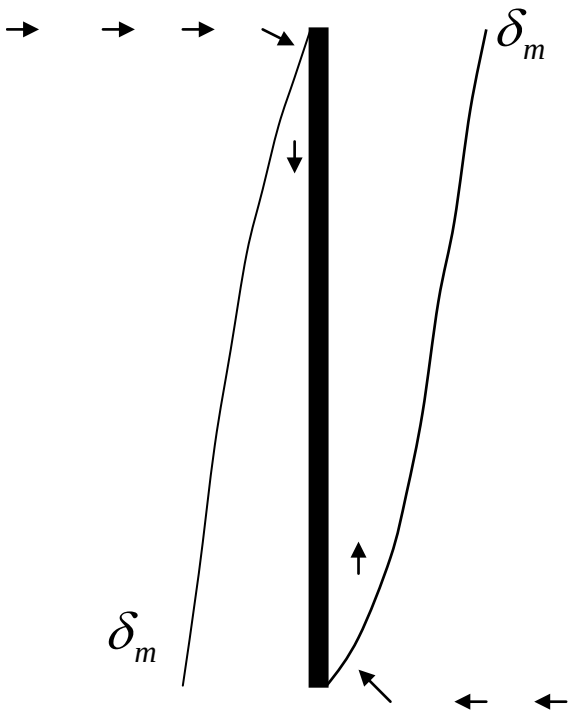
Consider a hot vertical surface present in a fluid. The surface is maintained at a temperature T_s, which is larger than the ambient temperature in the fluid T_e. Here is a sketch of the momentum boundary layer along the plate.



As shown in the sketch, the cold fluid rises along the plate surface, becoming heated in the process, and the momentum boundary layer grows in thickness with distance along the plate. A sample velocity profile in the momentum boundary layer is shown. Note that in this type of boundary layer, the velocity must be zero not only at the solid surface, but also at the edge of the boundary layer. Because the profile was sketched free-hand in PowerPoint, I am unable to show the smooth approach to zero velocity with a zero slope at the edge of the boundary layer properly, but that is how the correct velocity profile would appear. Compare this velocity profile with that in a momentum boundary layer that forms on a flat plate when fluid approaches it with a uniform velocity U_∞. You should try to make a sketch of the thermal boundary layer on the same plate when the fluid is air, for example, and also when it is a viscous liquid with a Prandtl number that is large compared with unity.

Now, let us consider a typical window in a home on a winter day when the outside air is at $10^\circ F$ and the inside of the room is at a balmy $68^\circ F$. What will the momentum boundary layers on

either side of the window look like? Try to sketch them yourself before looking at the figure. The arrows in the figure show the direction of air flow at the location where the air enters the boundary layer on the inside as well as on the outside, and the direction of air flow within the boundary layer. There is a slight transverse flow in each boundary layer, but on the scale of the picture, it is difficult to use the arrows to show it; therefore, I have drawn the flow in the boundary layers as being vertically downward or upward as appropriate.



What will the thermal boundary layers look like? Try sketching them. Also, you should make a sketch of the temperature distribution along the interior and exterior surfaces of the window from the bottom to the top. Will this permit you to explain why ice forms in a certain pattern on the outside surface of a window on really cold nights?

The Grashof and Rayleigh Numbers

In natural convection situations, an important dimensionless group is the Grashof number. To provide some physical significance to this group prior to defining it, we use a simple order of magnitude estimate of the natural convection velocity in the above examples. When fluid with a density ρ moves at a velocity V, the kinetic energy per unit volume can be written as $\frac{1}{2}\rho V^2$. This must come from some other form of energy, namely, potential energy lost by the fluid. Over a vertical distance L, the difference in potential energy between the less dense fluid in the boundary layer and the more dense fluid outside it can be approximately expressed as $g\,\Delta\rho\,L$, where g is the magnitude of the acceleration due to gravity, and $\Delta\rho$ is a characteristic density

difference between the boundary layer fluid and that far away. We can equate these two order of magnitude estimates, and neglect the factor of $1/2$, because this is only an order of magnitude analysis.

$$\rho V^2 \approx g \Delta\rho L$$

Therefore, a typical order of magnitude of the velocity arising from natural convection is

$$V \approx \sqrt{\frac{\Delta\rho}{\rho} g L}$$

Let us define a Reynolds number for the flowing fluid using this order of magnitude estimate.

$$\mathrm{Re}_L = \frac{LV}{\nu} = \sqrt{\frac{\frac{\Delta\rho}{\rho} g L^3}{\nu^2}}$$

so that $\mathrm{Re}_L^2 = \dfrac{\frac{\Delta\rho}{\rho} g L^3}{\nu^2}$. This is a dimensionless group that occurs often in natural convection problems, and is given the name **Grashof Number**, abbreviated as Gr.

$$\boxed{Gr = \frac{\frac{\Delta\rho}{\rho} g L^3}{\nu^2}}$$

The coefficient of volumetric expansion of a fluid β is defined as

$\beta = \dfrac{1}{V}\left(\dfrac{\partial V}{\partial T}\right)_P = \rho \dfrac{\partial}{\partial T}\left(\dfrac{1}{\rho}\right)_p = -\dfrac{1}{\rho}\left(\dfrac{\partial \rho}{\partial T}\right)_P$ where V is the specific volume, T is the temperature of the fluid and P is its pressure. Therefore, we can write

$\dfrac{\Delta\rho}{\rho} = -\dfrac{\Delta T}{\rho}\left(\dfrac{\partial \rho}{\partial T}\right)_p = \beta \Delta T$ where we have used a minus sign in relating $\Delta\rho$ to ΔT because both are defined as being positive, and as temperature increases, density decreases.

We can finally rewrite the definition of the Grashof number as follows.

$$\boxed{Gr = \frac{\beta \Delta T g L^3}{\nu^2}}$$

The Grashof number is related to the Reynolds number, and in heat transfer, the Prandtl number plays a significant role. Therefore, in natural convection heat transfer, we encounter another dimensionless group, called the Rayleigh number, abbreviated by Ra, which is the product of the Grashof and Prandtl numbers.

$$Ra = Gr \times \mathrm{Pr} = \boxed{\frac{\beta \Delta T\, g\, L^3}{\nu \alpha}}$$

Here, α is the thermal diffusivity of the fluid. The Nusselt number in natural convection heat transfer situations is typically a function of the Rayleigh number, the Prandtl number, and aspect ratio parameters.

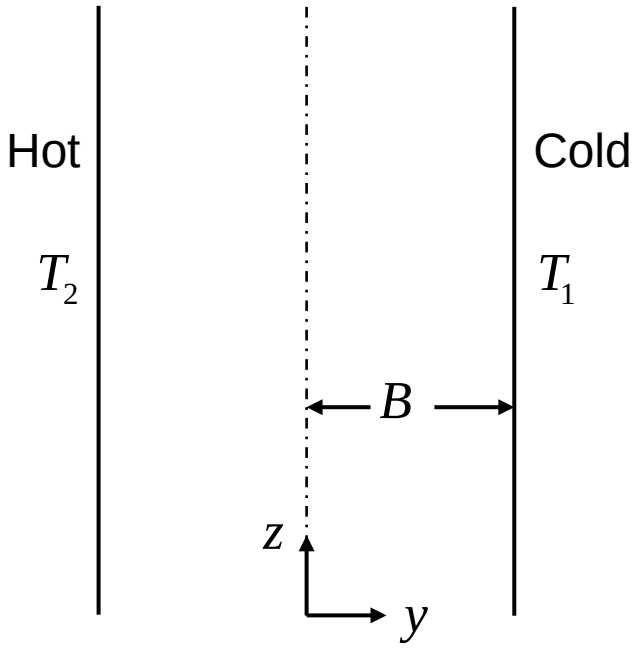
A variety of problems involving flows driven by buoyancy are considered in the book by Gebhart et al.

Reference

B. Gebhart, Y. Jaluria, R.L. Mahajan, and B. Sammakia, Buoyancy-Induced Flows and Transport, Hemisphere (1988).

Buoyancy Driven Flow Between Two Wide and Tall Parallel Plates

We analyze one of the simplest problems of flow driven by buoyancy. It is laminar flow that occurs between two wide and tall parallel plates as shown in the sketch. The problem is discussed in Bird, Stewart, and Lightfoot (2007) in Section 10.9. The treatment here is close to theirs, but based on the analysis given in Gebhart et al. (1988). We shall use the same notation as that in Bird et al. wherever possible. A sketch is given below.



The less dense fluid near the hot plate will rise and that near the cold plate will sink. In a closed container, there will be a turning flow near the top and bottom, and if it is sufficiently tall, the influence of this turning flow will be confined only to small regions near the top and bottom of the same length scale as the width between the plates. Away from these regions, we can consider the steady laminar flow as occurring only in the $z-$direction. This is the flow that we shall analyze.

Assumptions

1. Steady laminar Newtonian flow at constant viscosity μ. Also, assume constant thermal conductivity k and constant specific heat C_p.

2. The system is wide in the $x-$direction. Therefore, neglect edge effects $\Rightarrow$ $v_x \equiv 0;\ \frac{\partial \mathbf{v}}{\partial x} \equiv 0;\ \frac{\partial T}{\partial x} \equiv 0$.

3. Neglect end effects. Away from the top and bottom, $v_y \equiv 0$. Flow will occur only in the $z-$ direction. Also, the temperature field will be independent of z; that is ; $\frac{\partial T}{\partial z} \equiv 0$.

4. Make the Boussinesq approximation. This means that the density of the fluid is treated as a constant everywhere, except in the body force term.

5. Neglect viscous dissipation. There are no heat sources or sinks.

Continuity

$$\frac{\partial \cancel{v_x}^{\boxed{2}}}{\partial x} + \frac{\partial \cancel{v_y}^{\boxed{3}}}{\partial y} + \frac{\partial v_z}{\partial z} = 0$$

Therefore, v_z is independent of z. From assumption 1, it is independent of time, and from assumption 2, it is independent of x. Therefore, we conclude that $v_z = v_z(y)$ only.

Navier-Stokes Equations

$x-$ **component**

From assumption 2, and from the fact that the gravity vector is oriented in the negative $z-$ direction, so that $g_x = 0$, we obtain

$$\frac{\partial p}{\partial x} = 0$$

$y-$ **component**

From assumption 3, and from the fact that the gravity vector is oriented in the negative $z-$ direction, so that $g_y = 0$, we obtain

$$\frac{\partial p}{\partial y} = 0$$

Given the assumption of steady state, the pressure therefore depends only on z.

$z-$ **component**

$$\rho\left(\cancel{\frac{\partial v_z}{\partial t}}+\cancel{v_x}\frac{\partial v_z}{\partial x}+\cancel{v_y}\frac{\partial v_z}{\partial y}+v_z\cancel{\frac{\partial v_z}{\partial z}}\right)=-\frac{\partial p}{\partial z}+\rho g_z+\mu\left(\cancel{\frac{\partial^2 v_z}{\partial x^2}}+\frac{\partial^2 v_z}{\partial y^2}+\cancel{\frac{\partial^2 v_z}{\partial z^2}}\right)$$

1 2 3 continuity 2 continuity

Using the fact that $g_z=-g$, we obtain

$$\boxed{\mu\frac{d^2v_z}{dy^2}=\frac{dp}{dz}+\rho g}$$

where we have replaced the partial derivatives with ordinary derivatives because $v_z=v_z(y)$ only and $p=p(z)$ only. Because there is flow occurring here, the pressure variation can be different from the simple hydrostatic variation that would occur in the absence of flow. Later, we shall see that it is indeed hydrostatic, corresponding to the average density of the fluid in the space between the plates.

To make further progress, we turn to the energy equation for the temperature field.

$$\rho C_p\left(\cancel{\frac{\partial T}{\partial t}}+\cancel{v_x}\frac{\partial T}{\partial x}+\cancel{v_y}\frac{\partial T}{\partial y}+v_z\cancel{\frac{\partial T}{\partial z}}\right)=k\left(\cancel{\frac{\partial^2 T}{\partial x^2}}+\frac{\partial^2 T}{\partial y^2}+\cancel{\frac{\partial^2 T}{\partial z^2}}\right)+\mu\,\cancel{\Phi_v}+\cancel{S}$$

1 2 3 3 2 3 5 5

Therefore, the temperature field satisfies

$$\boxed{k\frac{d^2T}{dy^2}=0}$$

wherein we have replaced the partial derivative with the ordinary derivative, because the temperature field is independent of time (assumption 1), the x-coordinate (assumption 2), and the $z-$coordinate (assumption 3), implying $T=T(y)$ only.

The boundary conditions are:

$T(-B)=T_2$ Prescribed temperature on the left wall

$T(B)=T_1$ Prescribed temperature on the right wall

The solution for the linear temperature field between the two plates can be written immediately.

$$\boxed{T = \bar{T} - \frac{\Delta T}{2}\frac{y}{B}}$$

where $\bar{T}$ is the arithmetic mean temperature of the two plates given by $\bar{T} = \frac{1}{2}(T_1 + T_2)$, and $\Delta T = T_2 - T_1$.

Now that we have the temperature field, we can go back to the $z-$momentum equation after expressing the density of the fluid as a function of temperature. For this, we shall expand the density in a Taylor series about the mean temperature $\bar{T}$.

$$\rho = \bar{\rho} + \left(\frac{\partial \rho}{\partial T}\right)_{p, T=\bar{T}} \left(T - \bar{T}\right) + \ldots$$

where terms that are quadratic and higher order in the temperature difference $\left(T - \bar{T}\right)$ are represented by …, and we shall neglect them. This is a good assumption for relatively small values of ΔT. Note that the partial derivative of density with temperature is taken holding the pressure constant, and this derivative is then evaluated at the mean temperature.

We define the coefficient of volume expansion β as

$\beta = \frac{1}{V}\left(\frac{\partial V}{\partial T}\right)_p = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p$ where V is the specific volume of the fluid and is related to density through $\rho V = 1.$

Therefore, $\left(\frac{\partial \rho}{\partial T}\right)_{p, T=\bar{T}} = -\bar{\rho}\bar{\beta}$ where the overbar designates evaluation at $T = \bar{T}$, and the result for the density as a function of temperature becomes

$$\rho \approx \bar{\rho} - \bar{\rho}\bar{\beta}\left(T - \bar{T}\right)$$

The result is approximate, because we have neglected higher order terms in the Taylor series. Now, substitute for $T - \bar{T}$ from the solution for the temperature field, and insert the expression for the density in the $z-$momentum equation to yield the following ordinary differential equation.

$$\mu \frac{d^2 v_z}{dy^2} = \frac{dp}{dz} + \bar{\rho}\, g\left(1 + \bar{\beta}\,\frac{\Delta T}{2}\frac{y}{B}\right)$$

Differentiate this equation once with respect to y to eliminate the pressure gradient, yielding the third order differential equation

$$\boxed{\frac{d^3 v_z}{dy^3} = \frac{\bar{\rho}\bar{\beta} g \Delta T}{2B\mu}}$$

The expression on the right side is a constant. Therefore, the third derivative of the velocity component v_z is a constant. Thus, we should get a cubic function in y as the solution for the velocity field.

$$v_z(y) = c_0 + c_1 y + c_2 y^2 + \frac{\bar{\rho}\bar{\beta} g \Delta T}{12 B \mu} y^3$$

where $c_0, c_1,$ and c_2 are constants of integration that need to be determined. We need three conditions to evaluate them. The two no slip boundary conditions are

$v_z(-B) = 0$

$v_z(B) = 0$

These yield

$$c_0 + c_1 B + c_2 B^2 + \frac{\bar{\rho}\bar{\beta} g \Delta T}{12 \mu} B^2 = 0$$

$$c_0 - c_1 B + c_2 B^2 - \frac{\bar{\rho}\bar{\beta} g \Delta T}{12 \mu} B^2 = 0$$

so that $c_1 = -\frac{\bar{\rho}\bar{\beta} g \Delta T}{12 \mu} B$, and $c_0 = -c_2 B^2$. Hence, we can rewrite the solution for the velocity field as follows.

$$v_z(y) = \frac{\bar{\rho}\bar{\beta} g \Delta T B^2}{12 \mu} \left(\frac{y^3}{B^3} - \frac{y}{B} \right) + c_2 \left(y^2 - B^2 \right)$$

The third condition is obtained by noting that the mass flow rate across any section at an arbitrary value of z must be zero.

$$\int_{-B}^{B} \rho v_z(y) \, dy = 0$$

In applying this condition, when the result from the Taylor series for ρ is used along with the solution for the temperature field, we obtain

$$\rho = \bar{\rho} + \frac{\bar{\rho}\bar{\beta}\Delta T}{2} \frac{y}{B} = \bar{\rho} \left(1 + \frac{\bar{\beta}\Delta T}{2} \frac{y}{B} \right)$$

Therefore,

$$\int_{-B}^{B} \rho v_z(y)\,dy = \int_{-B}^{B} \bar{\rho}\left(1+\frac{\bar{\beta}\Delta T}{2}\frac{y}{B}\right)\left[\frac{\bar{\rho}\bar{\beta}g\Delta T B^2}{12\mu}\left(\frac{y^3}{B^3}-\frac{y}{B}\right)+c_2\left(y^2-B^2\right)\right]dy$$

$$= \bar{\rho}\left[-c_2\left(\frac{4}{3}B^3\right)+\frac{\bar{\beta}\Delta T}{2}\left\{-\frac{\bar{\rho}\bar{\beta}g\Delta T B^3}{45\mu}\right\}\right]$$

and setting the integral in the left side equal to zero yields

$$\boxed{c_2 = -\frac{\bar{\rho}\bar{\beta}^2 g\Delta T^2}{120\mu}}$$

Because we only retained terms up to and including $O(\Delta T)$ in the Taylor series, the above result for $c_2 \sim O(\Delta T^2)$ is considered negligible. Therefore, to within the order of our approximation, $c_2 = 0$. The final solution for the velocity field is cast in dimensionless form below to illustrate the physical significance.

$$\phi(Y) = \frac{B v_z \bar{\rho}}{\mu} = \frac{\mathrm{Gr}}{12}\left(Y^3 - Y\right)$$

Here, the dimensionless coordinate $Y = y/B$, and the dimensionless group Gr is defined as follows.

$$\boxed{\mathrm{Gr} = \frac{\bar{\rho}^2\bar{\beta}gB^3\Delta T}{\mu^2}}$$

Pressure distribution

We can evaluate the pressure gradient in the fluid by substituting the velocity distribution into the governing equation for $v_z(y)$. We find that $\mu\frac{d^2 v_z}{dy^2} = \frac{\bar{\rho}\bar{\beta}g\Delta T}{2B}y$. This yields $\frac{dp}{dz} = -\bar{\rho}g$. Therefore, the pressure variation is indeed hydrostatic, and corresponds to that in a fluid of uniform density $\bar{\rho}$.

References

1. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, Transport Phenomena, Wiley, 2007.

2. B. Gebhart, Y. Jaluria, R.L. Mahajan, and B. Sammakia, Buoyancy-Induced Flows and Transport, Hemisphere (1988).

Physical Significance of the Grashof Number

We can understand the significance of the Grashof number by considering two ratios, one being Gr/Re, and the other being $\mathrm{Gr}/\mathrm{Re}^2$.

First, consider the ratio Gr/Re. Let us use a length scale B and a velocity scale V for defining the Reynolds number as

$$\mathrm{Re} = \frac{BV\rho}{\mu}$$

It is understood that the properties ρ and μ are constants, and are evaluated at some mean temperature of the fluid.

Using a characteristic temperature difference ΔT, The Grashof number can be defined as follows.

$$Gr = \frac{\rho^2 \beta g B^3 \Delta T}{\mu^2}$$

Here, β is the coefficient of volumetric expansion of the fluid, and g is the magnitude of the acceleration due to gravity.

Now, consider first the ratio Gr/Re.

$$\frac{\mathrm{Gr}}{\mathrm{Re}} = \frac{\rho\beta g B^2 \Delta T}{\mu V} = \frac{\Delta\rho g B}{\mu\left(V/B\right)} \approx \frac{\text{Buoyant Force}}{\text{Viscous Force}}$$

The ratio $\mathrm{Gr}/\mathrm{Re}^2$ can be recast as follows.

$$\frac{\mathrm{Gr}}{\mathrm{Re}^2} = \frac{\mathrm{Gr}}{\mathrm{Re}} \times \frac{1}{\mathrm{Re}} = \frac{\Delta\rho g B}{\mu\left(V/B\right)} \frac{\mu\left(V/B\right)}{\rho V^2} \approx \frac{\text{Buoyant Force}}{\text{Inertia Force}}$$

Thus, in a flow situation that involves forced and free convection, these ratios can be used to evaluate the relative importance of the buoyant force when compared with the viscous or the inertia force. The Grashof number by itself represents the following ratio of forces.

$$\boxed{Gr \approx \frac{\text{Buoyant Force} \times \text{Inertia Force}}{\left(\text{Viscous Force}\right)^2}}$$

Surface-Tension-Driven Flows

Earlier, we learned about buoyancy-driven flows, also known as "natural convection." Another type of natural convection is that driven by interfaces between fluids. Consider a pool of a single-component liquid in a container. If the temperature is uniform along the free surface, the surface tension will be uniform as well. On the other hand, if a variation of temperature exists along the surface, then the surface tension, being a function of temperature, will vary along the free surface. In a pure component fluid, the surface tension decreases with increasing temperature. In a mixture, it is possible for the surface tension to increase with increasing temperature over some range of temperatures.

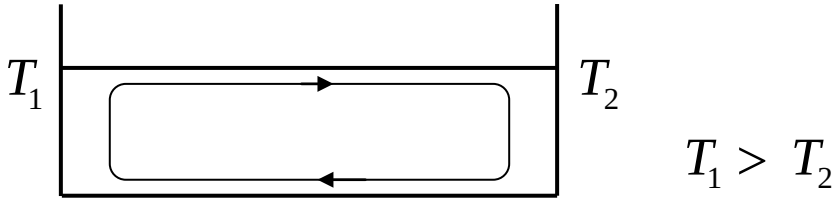
You may recall the tangential stress balance which reads as follows when there is no surface tension gradient.

$$\tau_{t_I} = \tau_{t_{II}}$$

Here, τ_t is the tangential component of the stress vector $\boldsymbol{t}$ acting on the surface, and I and II refer to the two fluid phases in contact, with the sign convention being that the normal vector points into fluid I. If there is a surface tension gradient $\nabla_s \sigma$ along the free surface, the tangential stress balance must be modified to read

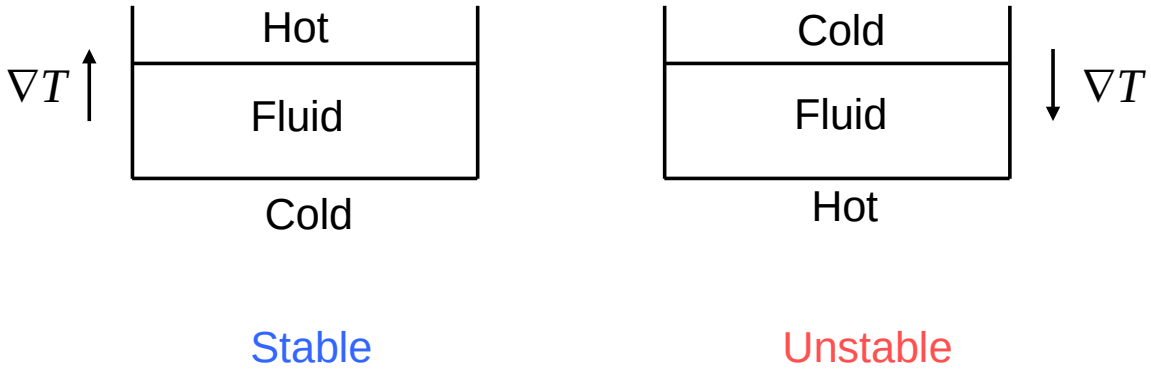
$$\tau_{t_I} - \tau_{t_{II}} = -\nabla_s \sigma$$

It is this inhomogeneity in the tangential stress balance that will lead to non-zero velocity fields in the adjoining fluids. The sketch below illustrates a typical circulation pattern that is set up by surface tension gradients in a pool of a pure component liquid.



It is assumed here that the temperature of the left boundary T_1 is larger than the temperature of the right boundary T_2. Because the surface tension decreases with increasing temperature, the surface tension is larger at the right boundary. If we consider a small element of fluid somewhere on the surface, it will experience a larger force pulling it toward the right than that pulling it toward the left. It is this differential force acting on surface elements that exerts traction on the adjoining liquid, causing it to move as shown. Of course, there will be a flow of the neighboring air as well, but it is not shown in the sketch.

If the temperature gradient is in a direction normal to the interface, the temperature will be uniform along the surface and one might assume that there can be no motion driven by surface tension gradients; however, it so happens that there are always small disturbances occurring in any system. A small disturbance that brings relatively cooler fluid from the bulk to the surface will locally increase the surface tension, drawing fluid toward the location, and opposing the flow that originally brought interior fluid to the surface. On the other hand, if the surface is cool and the fluid underneath is warmer, then a small disturbance that brings warm fluid to the surface will lead to that fluid being drawn away and the flow will continue. Thus, we have to deal with the issue of stability in that case. Both viscous effects and thermal conduction will tend to stabilize the system; therefore, for a given fluid, a critical temperature gradient in the direction normal to the interface will need to be exceeded for motion to set in. This phenomenon is known as the Marangoni instability, and the onset of this type of instability was first analyzed by Pearson (1958). Pearson showed that the cellular flow that is initiated consists of warm fluid rising toward the surface at the center of a cell, cooling as it moves out, and cooler fluid returning into the bulk around the edge of the cell. The hexagonal cellular flow patterns observed in shallow layers of spermaceti heated from below and recorded by Bénard in 1900-1901 were initially thought by Lord Rayleigh to be driven by buoyancy, leading him to publish an analysis of the buoyancy driven instability problem. Later, as pointed out by Pearson, it was recognized that the cellular flow in Bénard's experiments was really driven by a surface-tension driven instability. To learn more about the stability problem, two good books are available: Chandrasekhar (1981), and Koschmieder (1993). The stable and unstable cases are illustrated below.



Note that the orientation of the liquid film with respect to gravity is irrelevant. We can turn the picture upside down or sideways, and the instability will occur in the situation on the right, where relatively cooler fluid is present on the surface. Of course, in exceptional situations where the surface tension increases with increasing temperature, we would reverse the two pictures. Also, note that surface tension depends on composition at the interface. Therefore, analogous flows can be driven by composition gradients as well. This happens in mass transfer situations.

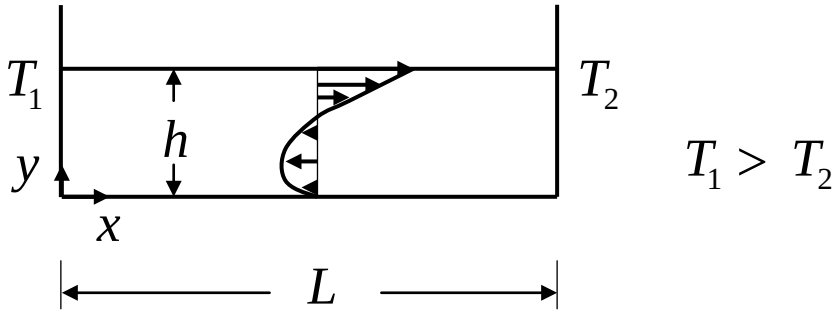
The relative importance of interfacial tension gradients versus buoyancy in driving natural convection has been discussed by Ostrach (1977), who defines a dynamic analog of the Bond number as follows.

$$Bo_{dynamic} = \frac{\rho g L^2}{|\sigma_T| \Delta T}$$

Here, ρ is the density of the fluid, and σ_T is the rate of change of surface tension with temperature. The symbol L represents a characteristic length scale, while ΔT is a characteristic temperature difference, and g is the magnitude of the acceleration due to gravity. The physical significance of the dynamic Bond number is that it represents the relative importance of buoyant forces in a fluid when compared with the force arising from interfacial tension gradients. The smaller the value of this group, the more dominant will be the role of surface tension gradients in causing motion. On Earth, in millimeter scale systems, surface tension forces dominate, but when experiments are performed in orbiting spacecraft or outer space probes, even on a length scale of 10 cm, surface tension forces will dominate in driving fluid motion, when compared with the role of the residual gravitational field.

Steady surface tension driven flow in a rectangular two-dimensional trough

Consider a rectangular trough containing a layer of liquid. If the trough is wide in the direction normal to the plane of the paper, we can treat the flow as two-dimensional, being uniform in the $z-$direction.



Levich (1962) analyzed the steady surface tension driven flow that arises in the above situation because of the steady temperature difference $T_1 - T_2$ along the free surface. Levich assumed that the heat transport was dominated by conduction so that the temperature field is linear between the two walls. He further postulated steady laminar Newtonian incompressible flow, and neglected end wall effects where the flow has to turn around. Away from the two walls, the flow is rather simple, with the velocity field given as follows.

$$v_x(y) = \frac{\sigma_T G h}{\mu}\left(\frac{3}{4}\frac{y^2}{h^2} - \frac{1}{2}\frac{y}{h}\right)$$

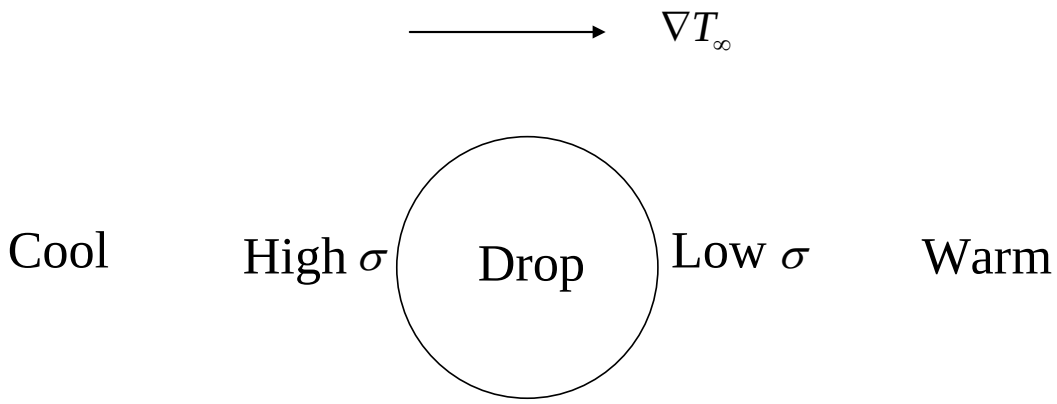
Here, $G = \frac{T_1 - T_2}{L}$ is the magnitude of the temperature gradient, and μ is the dynamic viscosity of the liquid. The velocity is maximum at the free surface as one might logically expect, and its value there is $\frac{\sigma_T G h}{4\mu}$. The velocity is zero at the bottom surface, and also at $y = 2h/3$.

When the flow is sufficiently rapid, convective heat transport, and inertia effects will need to be accommodated. The flow eventually achieves boundary layer structure. Such flows are discussed in detail in Carpenter and Homsy (1990) and references given therein.

Motion of Bubbles and Drops in a Temperature Gradient

A drop and a bubble are similar from a fluid mechanics perspective. A drop consists of a liquid, and is suspended in another liquid or a gas. A bubble contains gas or vapor, and is suspended in a liquid. In both cases, the interface is normally mobile, which means that fluid elements can move along the interface. This is in contrast to a rigid particle. In the following, the term "drop" is used generically to designate liquid drops and gas bubbles.

When a drop is placed in a continuous phase in which there is a temperature gradient, as depicted in the picture, the drop will move.



The reason for this motion can be traced to the variation of interfacial tension with temperature. Because of the temperature variation in the continuous phase, the temperature along the interface, and within the drop, will vary with position. In the figure, at the warm pole of the drop, the interfacial tension is normally smaller than that at the cold pole. The resulting gradient of interfacial tension along the interface leads to a discontinuity in the tangential stress across the interface. The tangential stress exerted on the continuous phase is in a direction such as to cause motion of this fluid toward the cold region. As a reaction, the drop is propelled in the direction of the temperature gradient, namely, toward warm regions. Because this motion is caused by temperature gradients, and involves the interface, it is called thermocapillary migration.

Young, Goldstein, and Block (1959) performed definitive experiments in which they applied a downward temperature gradient in a liquid column held between the anvils of a micrometer. Air

bubbles were introduced into the liquid, and the authors were able to hold them nearly motionless. They found that the temperature gradient required to hold a bubble nearly stationary was approximately proportional to its radius, consistent with their prediction. In fact, they analyzed the problem of the motion of a drop in the limit of negligible convective transport of momentum (zero Reynolds number) and energy (zero Peclet number), and predicted the thermocapillary contribution to the velocity of the drop to be

$$\boldsymbol{v} = \frac{2k\left(-\sigma_T\right)R\nabla T_\infty}{\left(2\mu+3\mu'\right)\left(2k+k'\right)}$$

Here, μ is the dynamic viscosity of the continuous phase and k is its thermal conductivity, and the primed quantities are the corresponding properties in the drop phase, σ_T stands for the rate of change of the interfacial tension with temperature, R is the radius of the drop, and ∇T_∞ is the applied temperature gradient. Because the interfacial tension decreases with temperature for single component fluids, the velocity of the drop points in the direction of the applied temperature gradient. As noted earlier, it is possible in the case of mixtures for the interfacial tension to increase with increasing temperature over a certain range of temperatures. This would lead to the motion of the drop toward relatively cooler regions in the continuous phase.

Young et al. (1959) solved a linear problem, and therefore, the contributions to the motion from thermocapillarity and gravity are additive. By summing the two contributions separately and setting the sum to zero, one can predict the temperature gradient necessary to hold a drop stationary.

An important dimensionless group in thermocapillary migration problems is the Peclet number for heat transfer. It is defined as

$$Pe = \frac{Rv_0}{\kappa}$$

where v_0 is a characteristic velocity and κ is the thermal diffusivity of the continuous phase. The characteristic velocity is estimated from the tangential stress balance (which is the motivating force for the motion of the drop) as

$$v_0 = \frac{\left|\sigma_T\right|\left|\nabla T_\infty\right|R}{\mu}$$

when this result is used for the characteristic velocity, the Peclet number is called the Marangoni number.

$$Ma = \frac{\left|\sigma_T\right|\left|\nabla T_\infty\right|R^2}{\mu\kappa}$$

The field of thermocapillary migration of drops and bubbles driven by a temperature gradient has matured substantially since the time of publication of the article by Young et al. particularly

because of the impetus of the space exploration program in the United States and abroad. In the reduced gravity environment aboard rockets, satellites, the Space Shuttle, and the International Space Station, phenomena such as this, which do not depend on gravity for their action, become prominent. You can learn more about the subject from Subramanian and Balasubramaniam (2001).

Rigid particles also will move when placed in a fluid in which the temperature varies with position. They will migrate in the direction opposite to that of the local temperature gradient. This phenomenon is known as thermophoresis. It is responsible for the deposition of soot on the interior of the glass wall of an oil lamp. Thermophoresis is important in combustion devices, for example. In a gas, a simplistic explanation is that the hotter molecules of gas impart a greater momentum on average to the warm side of the particle than the cooler molecules on the cool side, causing net momentum transfer in the direction of cooler fluid. In a liquid, where the molecules are not free to move about as in a gas, the explanation is based on the potential energy of attraction between the molecules of the liquid and the surface, and the gradient in pressure that develops along the surface of the particle as a consequence. An excellent discussion of thermophoresis as well as other "phoretic" phenomena (meaning motion driven by interfacial forces) can be found in a review by Anderson (1989).

References

J. Anderson, J., Colloid transport by interfacial forces, Ann. Rev. Fluid Mech. **21**, 61-99 (1989).

B.M. Carpenter and G.M. Homsy, High Marangoni number convection in a square cavity: Part II. Phys. Fluids **A2**, 137-149 (1990).

S. Chandrasekhar, Hydrodynamic and Hydromagnetic Stability, Oxford University Press, 1961, (reprinted by Dover in 1981)

E.L. Koschmieder, Bénard Cells and Taylor Vortices, Cambridge University Press, 1993.

V.G. Levich, Physicochemical Hydrodynamics, Prentice-Hall, 1962.

S. Ostrach, Low-gravity fluid flows, Ann. Rev. Fluid Mech. **14**, 313-345 (1982).

J.R.A. Pearson, On convection cells induced by surface tension, J. Fluid Mech. **4**, 489-500 (1958).

R. S. Subramanian and R. Balasubramaniam, The Motion of Bubbles and Drops in Reduced Gravity, Cambridge University Press, 2001.

N.O. Young, J.S. Goldstein, and M.J. Block, The motion of bubbles in a vertical temperature gradient, J. Fluid Mech. **6**, 350-356 (1959).

Part IX

Introduction to Mass Transport

Concentrations, Fluxes
Fick's Law
Conservation of Species
Boundary Conditions
Chemical Reactions

Introduction to Mass Transport

First, we need to develop some definitions. Please consult the textbook by Bird et al. for additional information.

Concentrations

ρ_i : Mass concentration of the i'th species is defined as

$$\rho_i = \frac{\text{Mass of Species } i}{\text{Unit Volume of Solution}}$$

c_i : Molar concentration of the i'th species is defined as

$c_i = \frac{\rho_i}{M_i}$ where M_i is the molecular weight of the i'th species. $\rho = \sum_i \rho_i$, $c = \sum_i c_i$.

Here ρ is the total mass density of the solution and c is the total molar density of the solution.

Velocities

$\boldsymbol{v}_i$: (vector) velocity of the i'th species in the laboratory reference frame

Mass average velocity $\boldsymbol{v}$

$$\boldsymbol{v} = \frac{\sum_i \rho_i \boldsymbol{v}_i}{\sum_i \rho_i} = \sum_i \frac{\rho_i}{\rho} \boldsymbol{v}_i = \sum_i \omega_i \boldsymbol{v}_i$$

Here, $\omega_i = \frac{\rho_i}{\rho}$ and is known as the mass fraction of the i'th species in the solution.

Molar average velocity $\boldsymbol{v}^*$

$$\boldsymbol{v}^* = \frac{\sum_i c_i \boldsymbol{v}_i}{\sum_i c_i} = \sum_i \frac{c_i}{c} \boldsymbol{v}_i = \sum_i x_i \boldsymbol{v}_i$$

Here, x_i is the mole fraction of the i'th species in the solution.

Fluxes

The (vector) mass flux of the i'th species in the laboratory frame of reference is $\boldsymbol{n}_i$.

$$\boldsymbol{n}_i = \rho_i \boldsymbol{v}_i$$

The (vector) mass flux of the i'th species in a reference frame moving at the instantaneous mass average velocity is $\boldsymbol{j}_i$, which is termed the diffusive flux.

$$\boldsymbol{j}_i = \rho_i \left(\boldsymbol{v}_i - \boldsymbol{v} \right) = \boldsymbol{n}_i - \rho_i \boldsymbol{v}$$

The individual diffusive fluxes $\boldsymbol{j}_i$ must add up to zero.

$$\sum_i \boldsymbol{j}_i = \sum_i \left(\rho_i \boldsymbol{v}_i - \rho_i \boldsymbol{v} \right) = \rho \boldsymbol{v} - \boldsymbol{v} \sum_i \rho_i = \boldsymbol{0}$$

Binary Systems – Fick's Law

The constitutive equation describing binary diffusion is known as Fick's law.

$$\boldsymbol{j}_A = - \rho \, D_{AB} \nabla \omega_A$$

where D_{AB} is known as the binary molecular diffusivity of A in B. For constant ρ, we can write

$$\boldsymbol{j}_A = - D_{AB} \nabla \rho_A$$

It is straightforward to show that $D_{AB} = D_{BA}$ by writing Fick's law for the mass flux of species B and combining the results for the two mass fluxes.

We can write the mass flux of A in the laboratory reference frame as

$$\boldsymbol{n}_A = \rho_A \boldsymbol{v} - \rho \, D_{AB} \nabla \omega_A$$

$$= \omega_A \left(\boldsymbol{n}_A + \boldsymbol{n}_B \right) - \rho \, D_{AB} \nabla \omega_A$$

Convective flux — Diffusive flux

In a like manner, the molar flux in the laboratory reference frame is

$$\boldsymbol{N}_A = c_A \boldsymbol{v}^* - c\, D_{AB} \nabla x_A$$

$$= x_A \left(\boldsymbol{N}_A + \boldsymbol{N}_B \right) - c\, D_{AB} \nabla x_A$$

Convective flux Diffusive flux

The diffusivity that appears in this result is the same as that which appears in the mass flux form of Fick's law.

Diffusive fluxes in systems containing more than two components are generally termed "multicomponent diffusive fluxes" and are described using the Stefan-Maxwell equations. For further information on multicomponent diffusion, you can consult Chapter 19 of the textbook by Bird et al. (1) as well as the book by Taylor and Krishna (2).

At room temperature, the binary diffusivity D_{AB} is typically of the order of $10^{-5}\ m^2/s$ in gases at atmospheric pressure, and $10^{-9}\ m^2/s$ in liquids. Techniques are available for predicting binary diffusivities from molecular parameters and are discussed in the textbook.

Analogy among molecular transport models

You may recall that when considering energy transport, in the discussion regarding the Prandtl number, we examined the analogy between momentum and energy transport in a one-dimensional context in which fluid flows in the $x-$direction with a velocity $v_x(y)$. In this case, Newton's law of viscosity for the momentum flux τ_{yx} (note that we are interpreting this symbol as the negative of the shear stress) can be written as

$$\tau_{yx} = -\mu \frac{\partial v_x}{\partial y}$$

and in a similar one-dimensional conduction problem, Fourier's law for the heat flux q_y can be written as

$$q_y = -k \frac{\partial T}{\partial y}$$

If we assume that the density and specific heat at constant pressure are constant, we can rewrite the above results in the following form.

x - momentum
concentration
$\Downarrow$

$$\tau_{yx} = -\nu \frac{\partial(\rho v_x)}{\partial y}$$

energy
concentration
$\Downarrow$

$$q_y = -\alpha \frac{\partial\left(\rho C_p \left\{T - T_{ref}\right\}\right)}{\partial y}$$

Likewise, we can write Fick's law in its one-dimensional form as

mass
concentration
$\Downarrow$

$$j_{Ay} = -\rho D_{AB} \frac{\partial \omega_A}{\partial y} = -D_{AB} \frac{\partial \rho_A}{\partial y}$$

Thus, we see that the molecular transport models all postulate a flux of an entity that is proportional to the concentration gradient of that entity.

Flux = - Transport Coefficient × Gradient of Concentration

Just as the Prandtl number, which is the ratio of the kinematic viscosity to the thermal diffusivity, permitted us to evaluate the relative rates of molecular transport of momentum and energy, a new ratio of the kinematic viscosity to molecular diffusivity, termed the Schmidt number, is useful in comparing relative rates of molecular transport of momentum and species. The abbreviation for the Schmidt number is Sc.

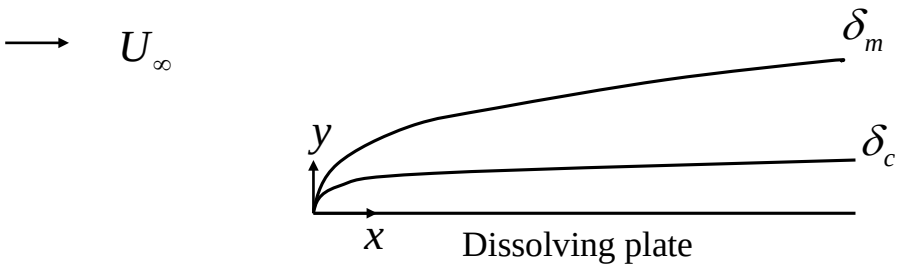
$$\mathrm{Sc} = \frac{\nu}{D_{AB}} = \frac{\text{Ability of a fluid to transport momentum by molecular means}}{\text{Ability of that fluid to transport species by molecular means}}$$

In gases, molecular transport of momentum and species occur by similar means, namely, by molecules moving from one place to another. While some momentum is transmitted through molecular interactions when two molecules come close to each other, the major contribution is from the movement of molecules themselves, which is the only mechanism for species transport

by molecular means. Therefore, Schmidt numbers in gases are typically of the order unity. In contrast, in a liquid, molecules are packed closely together, and diffusion is slow, as we know from the order of magnitude of diffusivities in liquids when compared with the order of magnitude in gases. On the other hand, momentum is efficiently transmitted in liquids through molecular interactions with each other. Therefore, Schmidt numbers in liquids are typically three orders of magnitude larger than those in gases.

$$\mathrm{Sc}_{\mathrm{liquid}} \sim O\left(10^3\right)$$

Thus, in flow over an object, the relative thicknesses of the momentum and concentration boundary layers reflect the magnitude of the Schmidt number, as the example given below for a liquid with a large Schmidt number $\mathrm{Sc} \gg 1 \quad \left(\nu \gg D_{AB}\right)$.



In gases, because the Schmidt number is close to unity, the two boundary layers are of comparable thicknesses.

Conservation of Species

The derivation of the equation of conservation of individual species is similar to the derivation of the equation of energy. We obtain the following conservation equation.

$$\frac{\partial \rho_i}{\partial t} + \nabla \bullet \boldsymbol{n}_i = r_i$$

Here, r_i is the rate at which the i'th species is produced per unit time per unit volume. The equation applies for each species in a multicomponent mixture. If the individual species conservation equations are added up, we obtain

$$\sum_{i=1}^{N} \frac{\partial \rho_i}{\partial t} + \sum_{i=1}^{N} \left(\nabla \bullet \boldsymbol{n}_i\right) = \sum_{i=1}^{N} r_i$$

or $$\frac{\partial \rho}{\partial t} + \nabla \bullet \boldsymbol{n} = 0$$

because the individual production rates must add up to zero net rate of mass production. Because $\boldsymbol{n} = \rho \boldsymbol{v}$, this becomes the continuity equation that was derived earlier.

If we confine attention to a binary system containing A and B, Fick's law can be used to relate the mass flux to the gradient of mass fraction of species A as follows.

$$\boldsymbol{n}_A = \rho_A \boldsymbol{v} - \rho D_{AB} \nabla \omega_A$$

Substituting this result into the conservation equation for species A, we obtain

$$\frac{\partial \rho_A}{\partial t} + \nabla \bullet \left(\rho_A \boldsymbol{v} \right) = \nabla \bullet \left(\rho D_{AB} \nabla \omega_A \right) + r_A$$

For constant density ρ and diffusivity D_{AB}, this can be simplified to the following form.

$$\boxed{\frac{\partial \rho_A}{\partial t} + \boldsymbol{v} \bullet \nabla \rho_A = D_{AB} \nabla^2 \rho_A + r_A}$$

This is usually the starting point in analyzing mass transport problems in binary systems where the stated assumptions hold. Let us compare this conservation equation with that of conservation of energy for negligible viscous dissipation and constant thermal diffusivity α.

$$\boxed{\frac{\partial T}{\partial t} + \boldsymbol{v} \bullet \nabla T = \alpha \nabla^2 T + \frac{S}{\rho C_p}}$$

The analogy is evident. Therefore, if a mass transport problem and a heat transport problem have the same initial and boundary conditions, the solutions for the concentration and temperature fields (suitably scaled) will be identical as well. This also means that we can perform mass transport experiments to infer information about the corresponding heat transport problem and vice versa.

Boundary Conditions in Mass Transport

In considering mass transport of species between two phases, there is the possibility that some species may adsorb on the interface. Such species are called surface-active species or surfactants. We shall exclude them from discussion in this elementary approach to writing boundary conditions in mass transport problems.

The first type of boundary condition arises from the assumption that thermodynamic equilibrium prevails between the two phases at the interface. Therefore, the chemical potential of each species must be the same on either side of the interface. In practice, we do not work with chemical potentials, but rather with concentrations of species expressed in a variety of units, such as mole fractions, mass fractions, molar concentrations, and mass concentrations. Therefore, we can write the requirement of equilibrium at the interface in the form that the concentrations of each species on either side of the interface must be in equilibrium.

A second condition we can write at an interface is that the mass flux of each species normal to the interface must be continuous across the interface. If we assume that the interface is stationary, this simplifies to the statement

$$\boldsymbol{n}_{A,\,I} \bullet \boldsymbol{n} = \boldsymbol{n}_{A,\,II} \bullet \boldsymbol{n}$$

where the subscript A refers to species A, and the subscripts I and II refer to phases I and II, respectively. The unit normal $\boldsymbol{n}$ is oriented normal to the interface, and by convention, points into phase I. The same condition applies to each individual species in the mixture.

At a fluid-fluid interface, where we are solving for the concentration fields in both phases, we must write both the equilibrium condition and the statement of continuity of normal flux. At a solid-fluid interface, typically we are not interested in describing diffusion through the solid, so that only the equilibrium condition is used when solving for the concentration field in the fluid.

Chemical Reactions in Mass Transport Problems

Chemical reactions are classified in general as being either homogeneous or heterogeneous. In a continuum context, a homogeneous reaction takes place in every volume element in a continuum, and the rate of production of species i per unit volume per unit time is represented by the symbol r_i in mass units and R_i in molar units. The reaction rate can depend on position and time, and is usually obtained from a model of the kinetics. Disappearance of species i by homogeneous reaction simply implies a negative value of r_i.

A homogeneous reaction does not affect the boundary conditions. In contrast, when a reaction is heterogeneous, it occurs on a bounding surface in the domain, and does not lead to a production/removal term in the governing conservation equation. Instead, the boundary condition at the surface matches the flux of species i to the surface with the rate at which species i is being consumed by chemical reaction at the surface. A good example of the way a heterogeneous chemical reaction is accommodated can be found in the worked out Example 18.3.1 in the textbook by Bird et al. In this example, the authors assume a pseudo-first-order reaction at a catalyst surface, leading to the disappearance of species A. The flux of A to the surface, in the $z-$direction, labeled N_{Az} is written as

$$N_{Az} = k\, c\, x_A$$

Here, k is a pseudo-first-order rate constant for the disappearance of A, c is the total molar concentration in the fluid at the surface, and x_A is the mole fraction of A in the fluid at the surface. The fluid happens to be a gas in this example.

In the problem considered in the main part of Section 18.3, a heterogeneous reaction occurs at a catalyst surface, but the rate of reaction is so large that Bird et al. assume that the reaction is instantaneous. In the context of the above boundary condition this implies that one must take the limit as $k \to \infty$. In this limit, for a finite molar flux of A to the surface, we must require that the mole fraction of A vanish at the solid surface.

Interestingly, in dealing with problems involving a porous catalyst, it is often convenient to use a pseudo-homogeneous model of transport of species in the catalyst. This problem is considered in Section 18.7 in the textbook by Bird et al. In this approach, no attempt is made to account for the fine-grain structure of the porous material, which consists of tortuous passages in a solid. Instead, the porous medium is treated as a single continuum. How do we accommodate a heterogeneous chemical reaction in this case? It is done through the use of a pseudo-homogeneous reaction approach, introducing a reaction term in the governing conservation equation for the continuum. For more details, you should study the worked-out problem in Section 18.7.

References

1. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, Transport Phenomena, Wiley, 2007.

2. R. Taylor and R. Krishna, Multicomponent Mass Transfer, Wiley, 1993.

Part X

Simple Example Problems in Mass Transport

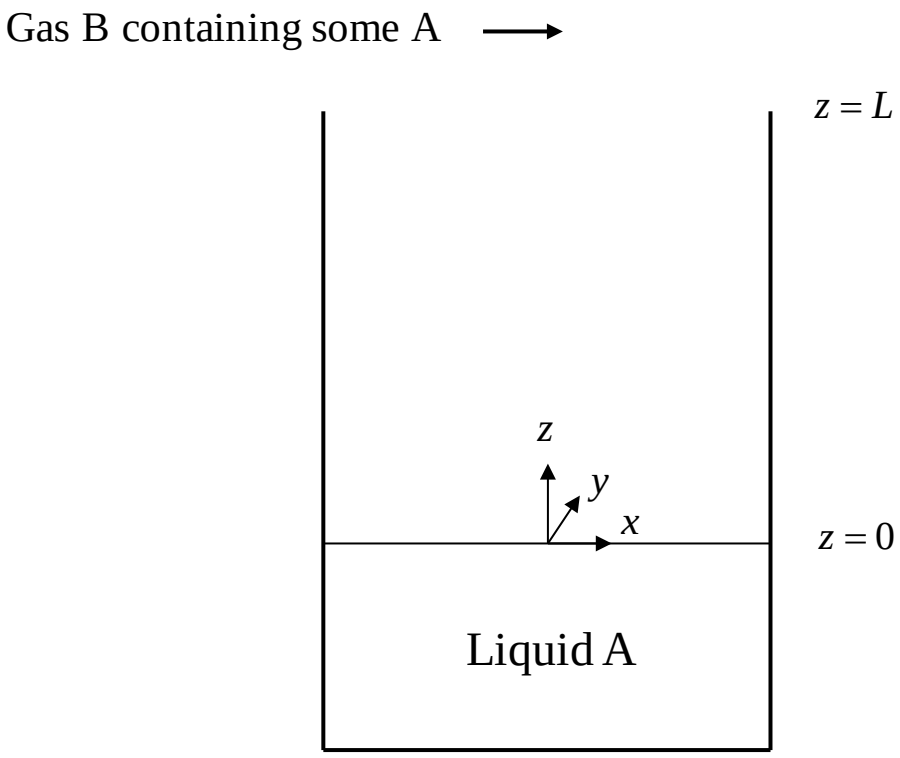
Evaporating Layer of Liquid

Mass Transfer Between a Sphere and an Unbounded Fluid

Evaporating Layer of Liquid

The one-dimensional transport of one gaseous species through another non-transferring species is a useful example from which one can learn about the important role played by convective transport that arises from diffusion. This contribution is absent in the analogous one-dimensional conduction problem. The present example is discussed in Section 18.2 of the textbook by Bird et al., and is somewhat incorrectly labeled as "Diffusion through a stagnant gas film." In fact, there is motion in the "gas film" as we shall see, and the correct title should have been the one noted in the first sentence of this paragraph.

The sketch below illustrates the experiment that we plan to model.



A small stationary pool of liquid is present in a container. The liquid is sufficiently volatile that it exerts a significant vapor pressure at the prevailing room temperature. A gentle flow of air is maintained at the top so that the vapor is carried away by air. To keep the problem statement as general as possible, the air approaching the container is assumed to contain some vapor A and the remainder is non-transferring air (B). The liquid is assumed to be already saturated with air, so that there is no net transport of air through the air-vapor mixture above the liquid layer.

It is assumed that the evaporation rate is sufficiently small that the mole fraction of A in the air stream remains virtually unchanged during its transit over the top of the container, and we label

this mole fraction x_{A2}. At the surface of the liquid, equilibrium is assumed between the gas and the liquid phases, so that the partial pressure of the vapor A in the gas phase at the interface is equal to the vapor pressure of the liquid at the prevailing temperature. We assume the gas mixture to be ideal and write

$$x_A(0) = x_{A1} = \frac{P_A}{P_t}$$

where P_A is the equilibrium vapor pressure of A and P_t is the total pressure in the gas, assumed uniform throughout the system.

At steady state, the vapor A diffuses through the gas column because the mole fraction of A at the liquid-gas interface is larger than that at the top of the container. As the liquid continues to evaporate, the height of the liquid column will gradually go down, and this change can be monitored with suitable optics. By measuring the rate of evaporation of the liquid, we expect to be able to calculate the diffusivity of the vapor A in the non-transferring gas B. In order to do this, we construct a model of transport in the gas column above the liquid surface.

The following assumptions are made to keep the problem simple, and yet retain the essential features.

1. Neglect the change of liquid height with time; for slowly evaporating liquids, this is a good assumption.

2. Steady one-dimensional transport in the $z-$direction; neglect gradients of concentration in the lateral directions x and y.

3. There are no chemical reactions in this system.

4. The temperature and pressure are uniform; this means that the molar concentration in the gas phase is uniform.

5. The binary diffusivity D_{AB} is constant.

Because the molar concentration is uniform, it is sensible to use the molar form of the conservation equation for species A. In rectangular Cartesian coordinates,

$$\frac{\partial c_A}{\partial t} + \frac{\partial N_{Ax}}{\partial x} + \frac{\partial N_{Ay}}{\partial y} + \frac{\partial N_{Az}}{\partial z} = R_A$$

so that $\frac{dN_{Az}}{dz} = 0$

This means that $\boxed{N_{Az} = \text{constant}}$ everywhere, because from assumption 1, the flux is independent of time and position coordinates x and y. Of course, we do not know the value of this constant molar flux of A. To determine its value, we must proceed to use the component of Fick's law in the $z-$direction.

$$N_{Az} = x_A\left(N_{Az} + N_{Bz}\right) \quad - \quad c\,D_{AB}\frac{dx_A}{dz}$$

Convective flux Diffusive flux

We had assumed that species B (air) is non-transferring. This means that the steady flux of species B, $N_{Bz} = 0$. As a result, Fick's law can be simplified to the following form.

$$\frac{1}{1-x_A}\frac{dx_A}{dz} = -\frac{N_{Az}}{cD_{AB}} = C_1, \text{ where } C_1 \text{ is a constant.}$$

In the analogous one-dimensional heat transport problem, there would be no convective transport term, and the gradient of temperature in the $z-$direction is simply a constant, leading to a linear temperature profile.

Integrating Fick's law, we find

$$-\ln\left(1-x_A\right) = C_1 z + C_2$$

where C_2 is a constant of integration. By applying the boundary conditions

$x_A(0) = x_{A1}$

$x_A(L) = x_{A2}$

the two constants can be evaluated. We find

$$C_2 = -\ln\left(1-x_{A1}\right) \qquad C_1 = \frac{1}{L}\ln\left(\frac{1-x_{A1}}{1-x_{A2}}\right)$$

Substituting in the solution,

$$-\ln\left(1-x_A\right) = \frac{z}{L}\ln\left(\frac{1-x_{A1}}{1-x_{A2}}\right) - \ln\left(1-x_{A1}\right)$$

which can be rewritten in the following convenient form.

$$\boxed{\frac{1-x_A}{1-x_{A1}} = \left(\frac{1-x_{A2}}{1-x_{A1}}\right)^{z/L}}$$

or equivalently as

$$\boxed{\frac{x_B}{x_{B1}} = \left(\frac{x_{B2}}{x_{B1}}\right)^{z/L}}$$

Recall that we are interested in the flux at the liquid surface. We know from the solution that the flux N_{Az} is independent of position z in the column and is equal to $-cD_{AB}\,C_1$.
Therefore,

$$N_{Az} = -\frac{c\,D_{AB}}{L}\ln\left(\frac{1-x_{A1}}{1-x_{A2}}\right) = -\frac{c\,D_{AB}}{L}\ln\left(\frac{x_{B1}}{x_{B2}}\right)$$

This can be recast as

$$\boxed{N_{Az} = \frac{c\,D_{AB}}{L\,x_{Bm}}\left(x_{A1} - x_{A2}\right)}$$

where

x_{Bm} is the log mean of the concentration of B over the diffusion path.

$$\boxed{x_{Bm} = \frac{x_{B2} - x_{B1}}{\ln\left(\frac{x_{B2}}{x_{B1}}\right)}}$$

In a dilute system, the mole fraction of A will be small compared with that of B everywhere. This means that we can write $x_{Bm} \approx 1$ in the result for the flux, so that the flux in a dilute system is approximated by

$$N_{Az} \approx \frac{c\,D_{AB}}{L}\left(x_{A1} - x_{A2}\right) = D_{AB}\frac{c_{A1} - c_{A2}}{L}$$

which is analogous to the result one would get in the one-dimensional conduction problem. If we examine the origin of the appearance of x_{Bm} in the result for the molar flux, we see that it is a consequence of including the convection or "drift" due to diffusion of A in the mixture. Thus, we

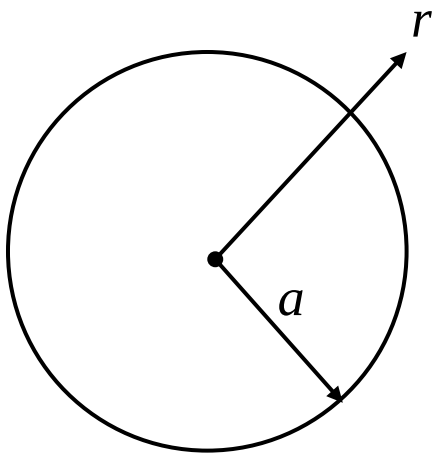
conclude that in a dilute system, the role of the convective contribution arising from diffusion can be considered negligible.

Because $x_{B\,m}$ must be less than unity, we see that the flux of A is enhanced by the convective drift arising from diffusion over that which would arise if this convection is ignored.

Try sketching the composition profiles of A and B in the gas above the liquid layer. We know that the mole fraction of A is relatively large at the liquid surface and decreases exponentially to the mole fraction prevalent in the stream of air at the top of the gas column. Because the two mole fractions must add to unity everywhere, the mole fraction of B must be relatively large at the top and decrease as one approaches the liquid surface. Therefore, there is clearly a gradient of composition of B. This must lead to a diffusive flux of B toward the liquid surface, yet we said there is no net flux of B. Is this a contradiction? Try to figure out why the net flux is zero, even though the diffusive flux of B is not zero.

Mass Transfer Between a Sphere and an Unbounded Fluid

When a single-component liquid drop evaporates into air, or when a solid, modeled as a single-component sphere, dissolves in a liquid or sublimes into a gas, we can construct a simple model of the diffusive transport that occurs between the object and the surrounding fluid. The model can help us calculate the rate of mass transfer, and eventually the rate of change of the radius of the sphere with time.



Assumptions

1. The sphere contains a pure component A; therefore, we need to consider the mass transport process only in the surrounding fluid.

2. The fluid is unbounded in extent and quiescent. It contains only the diffusing species A and a non-transferring species B.

3. The motion arising from diffusion can be neglected. This requires that either the mixture in the fluid be dilute in species A, consisting primarily of the non-transferring species B, or that the rate of mass transport be small.

4. The problem is spherically symmetric. This means that in a spherical polar coordinate system (r,θ,ϕ) there are no gradients in the polar angular coordinate θ, or in the azimuthal angular coordinate ϕ.

5. After an initial transient, steady state is assumed to prevail. This implies that the change in size of the sphere due to mass transfer occurs on a time scale that is very large compared with the time scale for the diffusion process for a given radius of the sphere to reach steady state.

6. There are no chemical reactions.

Subject to the above assumptions, the equation of conservation of mass for species A in the fluid phase in spherical polar coordinates is simplified as follows.

$$\boxed{5} \qquad \boxed{4} \qquad \boxed{4} \qquad \boxed{6}$$

$$\cancel{\frac{\partial c_A}{\partial t}} + \frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2 N_{Ar}\right) + \frac{1}{r\sin\theta}\cancel{\frac{\partial}{\partial \theta}}\left(N_{A\theta}\sin\theta\right) + \frac{1}{r\sin\theta}\cancel{\frac{\partial}{\partial \phi}}\left(N_{A\phi}\right) = \cancel{R_A}$$

Because there are no gradients in the θ and ϕ directions, and there is no time-dependence, the flux N_{Ar} depends only on r. Thus, we can write

$$\frac{1}{r^2}\frac{d}{dr}\left(r^2 N_{Ar}\right) = 0$$

Integration leads to the result

$r^2 N_{Ar} = C_1$, which can be recast as $\boxed{N_{Ar}(r) = \frac{C_1}{r^2}}$

where C_1 is an arbitrary constant of integration. Now, we proceed to use Fick's law.

$$N_{Ar} = x_A\left(N_{Ar} + N_{Br}\right) - c\,D_{AB}\frac{dx_A}{dr}$$

The first term in the right side corresponds to convective transport, which can be neglected in this problem because of assumptions 2 and 3. Thus, we obtain the following first order ordinary differential equation for the mole fraction of species A in the fluid.

$$\boxed{\frac{dx_A}{dr} = -\frac{C_1}{c\,D_{AB}}\frac{1}{r^2}}$$

Integration of this equation is straightforward, and leads to the following solution.

$$x_A(r) = \frac{C_1}{c\,D_{AB}}\frac{1}{r} + C_2$$

There are two arbitrary constants that need to be evaluated. Therefore, we must write two boundary conditions. At the surface of the sphere, we can assume equilibrium to prevail between the two phases. For example, if species A is evaporating into a gas, the partial pressure of species A in the gas phase at the interface can be assumed to be equal to its equilibrium vapor pressure at the prevailing temperature. If the gas mixture is assumed ideal, then the mole fraction of species A in the gas phase at the interface is the ratio between this equilibrium vapor pressure of A and

the prevailing total pressure in the gas phase. In non-ideal cases, a corresponding result can be used to obtain the equilibrium mole fraction of species A in the gas phase at the interface. Likewise, for a solid dissolving in a liquid, or subliming into a gas, the equilibrium mole fraction of species A in the fluid at the interface can be obtained.

$$x_A(a) = x_{A1}$$

Far from the sphere, we can assume the composition to approach that in the fluid in the absence of the sphere. Thus,

$$x_A(\infty) = 0$$

Application of these two boundary conditions permits us to evaluate the constants C_1 and C_2 as

$$\boxed{C_1 = c\, D_{AB}\, a\, x_{A1}} \qquad \boxed{C_2 = 0}$$

Substituting these results in the solution leads to the following result for the radial distribution of the mole fraction of species A in the fluid.

$$\boxed{\frac{x_A(r)}{x_{A1}} = \frac{a}{r}}$$

The flux of species A is given by

$$\boxed{N_{Ar}(r) = c\, D_{AB}\, a\, x_{A1}\, \frac{1}{r^2}}$$

so that the molar rate of transport at the surface of the sphere can be written as

$$W_A = 4\pi a^2\, N_{Ar}(a) = \boxed{4\pi\, c\, D_{AB}\, a\, x_{A1}} = \boxed{4\pi\, D_{AB}\, a\, c_{A1}}$$

where we have used the fact that the product $c\, x_{A1} = c_{A1}$, the molar concentration of A in the fluid at the interface. Assuming that the molar rate of transport is relatively small, we can use a mass balance on the sphere to deduce the rate of change of its size with time. Let the molecular weight of A be M_A, and the density of the sphere be ρ. Then, we can write

$$\frac{d}{dt}\left(\frac{4}{3}\pi a^3 \rho\right) = 4\pi a^2 \rho \frac{da}{dt} = -4\pi\, D_{AB}\, M_A\, a\, c_{A1}$$

which leads to a differential equation for the time-dependence of the radius of the sphere.

$$\boxed{a\frac{da}{dt} = -\frac{D_{AB} M_A c_{A1}}{\rho}}$$

If the radius at time zero is a_0, then the solution can be written as

$$\boxed{a^2(t) = a_0^2 - \frac{2 D_{AB} M_A c_{A1}}{\rho} t}$$

The Quasi-Steady State Assumption

Note that we assumed steady state to prevail in the diffusion problem, which, strictly speaking, requires the size of the sphere to remain unchanged. As stated in assumption 5, this only requires that the time scale over which the sphere changes size appreciably is large compared with the time scale over which the diffusion process around a sphere of constant size reaches steady state. Then, the rate of mass transfer from the sphere to the fluid can actually be used to calculate the time evolution of the size of the sphere. This type of assumption is called a quasi-steady state assumption. It is invoked commonly in transport problems where there are two very different time scales involved. You may have encountered the quasi-steady assumption in the problem of calculating the rate of change of the height of liquid in a large storage tank through a small pipe at the bottom. To calculate the velocity of flow out of the pipe and therefore the volumetric flow rate, we usually assume the level of the fluid in the tank to remain sensibly constant. After obtaining such a volumetric flow rate from a steady-state model, it can be used in an unsteady mass balance on the contents of the tank to calculate the rate of change of height of the liquid in the tank.

Reference

R.B. Bird, W.E. Stewart, and E.N. Lightfoot, Transport Phenomena, Wiley, 2007.

Part XI

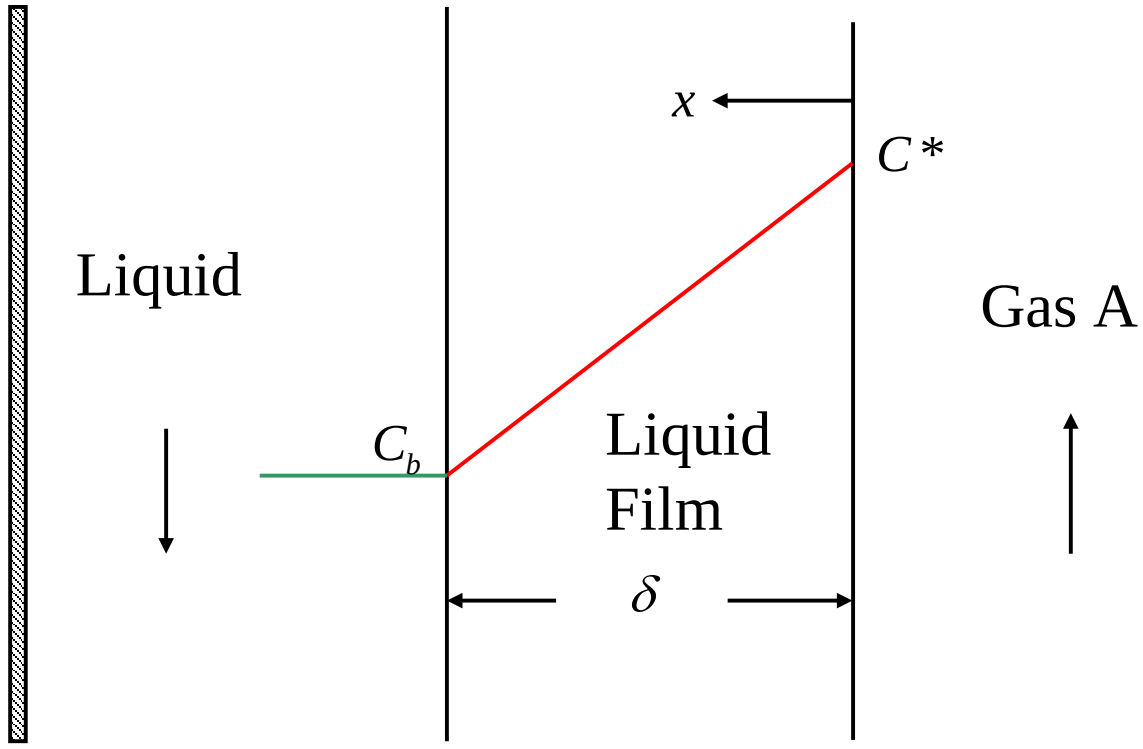
Film and Penetration Models

Film and Penetration Models

In many situations, mass transfer problems are complicated because of the geometry. Examples are absorption or distillation in a packed or tray column, liquid-liquid extraction, and adsorption and ion exchange. It is difficult to model these situations directly from first principles because the geometries are too complex to describe using simple coordinate systems, and also because of the multiphase nature of these problems. As a consequence, chemical engineers have developed empirical approaches for dealing with these mass transfer operations. Two of the most commonly used models are the "Film Model" and the "Penetration Model." The transport theory is done from first principles in these idealized "model geometries" and extended to the actual operation by empirical means. The film model is described first.

The Film Model

Consider, for example, gas absorption in a packed column. The solvent (B) is admitted at the top and flows down the column, presumably in the form of a thin film that covers the surface of the packing material. The gas, assumed here to be pure A, flows upward in the interstitial region in the packing where liquid is not present. Exchange of the species A occurs between the gas and the liquid, and the objective is to be able to predict mass transfer rates. If we consider the situation around a single particle of packing, and magnify it, we can represent it approximately by the sketch.



The surface of the packing particle is represented by the shaded region in the left. The approach in the film model is to assume that most of the liquid is well mixed with some bulk concentration of the dissolving solute (A) that is given by C_b. All the concentration change in the liquid is assumed to occur in a **stationary** film of liquid of thickness δ. This is a fictitious film. It does not really exist because the liquid is flowing down the side of the packing and there is some appropriate velocity distribution in the liquid film covering the packing surface. It certainly is not "stationary." This mass transfer "film" is not the liquid film, but an imaginary region in the vicinity of the phase interface. Its thickness, to be established shortly, is adjusted so that the flux of A predicted from this "film model" in a simple situation matches the experimentally observed flux in the absorption column on average. If the gas phase consists of an inert C and the dissolving species A, then we would postulate a similar "film" on the gas side of the interface as well.

The main idea in this model is that the fluid in the "film" is stationary; the fact that it is flowing is accommodated by fitting the "film thickness" to match experimental mass transfer rates. Diffusion occurs in the direction normal to the phase interface, and in this magnified picture, it occurs in a single direction, labeled x in the drawing. We assume equilibrium to exist between the two phases at the interface, leading to some equilibrium concentration C^* in the liquid at the interface. This equilibrium concentration corresponds to the prevailing temperature and pressure and the composition of the gas phase. At the other edge of the film, the concentration is assumed to be the uniform bulk concentration C_b as shown in the figure. Also shown in the figure is an idealized concentration profile within the film in the limit of low mass transfer rates. We shall see shortly why this is a straight line.

We begin with the equation of conservation of mass in rectangular Cartesian coordinates. Assuming constant physical properties, steady unidirectional transport, and no chemical reactions, leads to the following result for the flux of A in the x-direction.

$$\frac{dN_{Ax}}{dx} = 0$$

This means that the flux N_{Ax} is constant along the diffusion path. We have used the molar flux and molar concentrations in posing this problem, but we can equally well use the mass flux and mass concentrations. In this example, we assume low mass transfer rates (or a dilute solution), so that the drift in the fluid in the x-direction that occurs due to diffusion, and the resulting convective flux of A, can be neglected as a first approximation. If the concentration of A is low, the molecular weight of the mixture is approximately the same as that of the solvent, and remains constant throughout the diffusion path. In this situation, both the total mass density and the molar concentration are constant, and we can simplify Fick's law in molar units to

$$N_{Ax} = -D\frac{dC_A}{dx}$$

Because the variation of concentration is assumed to be one-dimensional, we have replaced the partial derivative with the ordinary derivative, and the symbol D stands for the binary diffusivity

of A in B. Knowing that N_{Ax} is a constant, we can integrate the above equation, along with the boundary conditions

$$C_A(0) = C^*$$
$$C_A(\delta) = C_b$$

to obtain the linear concentration profile given below as the solution.

$$\frac{C^* - C_A}{C^* - C_b} = \frac{x}{\delta}$$

This profile is displayed in the sketch. The molar flux of A is constant throughout the film and is given by

$$N_{Ax} = D\frac{C^* - C_b}{\delta} = k_c\left(C^* - C_b\right)$$

where we have defined a mass transfer coefficient k_c. Therefore, the mass transfer coefficient is related to the film thickness in this example as follows.

$$\delta = \frac{D}{k_c}$$

So, you can see that if you perform an experiment on a dilute system in a packed column at low mass transfer rates and obtain the mass transfer coefficient, you can estimate the fictitious film thickness corresponding to that experiment. Of course, this is an average for the entire column. If we cast this in terms of the Sherwood number $Sh = \frac{k_c L}{D}$, where L is a length scale used in the definition of the Sherwood number, then $\delta = \frac{L}{Sh}$. Therefore, if we determine the Sherwood number as a function of the Reynolds and Schmidt numbers, for example, we'd know the dependence of the fictitious film thickness in terms of these parameters as well.

Note that the mass transfer coefficient appears to be proportional to the first power of the diffusivity. This is not the case with other models nor is it always the case in experiments. To overcome this objection, proponents of the film model have suggested that the film thickness be allowed to depend on the diffusivity to a suitable power.

So far, the film model has not been **predictive**. That is, we have had to rely on experiments to infer the fictitious film thickness. But now, if we want to estimate the mass transfer rates in the same column for other conditions such as for high mass transfer rates (where the convective transport of A arising from the motion due to diffusion cannot be ignored), or a reacting system,

or a multicomponent system where we must use the Maxwell-Stefan equations, we can use the film model in a predictive manner. We can use the "known" film thickness inferred from experiments on a dilute non-reacting binary system in a film model of mass transport in which one or more of these additional effects are accommodated, and thereby predict the mass transfer rates.

Illustration of the use of the Film Model – Homogeneous Chemical Reaction

As an illustration, consider a case where we have the same situation as depicted in the sketch, but A is consumed in a first order chemical reaction. In this case, the rate of production of A per unit volume, R_A, can be written as $R_A = -k C_A$, where k is a rate constant. All the assumptions we already made apply here. Therefore, going through the simplification of the mass conservation equation for species A in the liquid film, and using Fick's law, leads to

$$D\frac{d^2C_A}{d\,x^2} - k\,C_A = 0$$

The boundary conditions remain the same. Permitting A to be present in the bulk liquid will cause an additional complication because the bulk concentration of A will depend upon the location in the main flow direction. Therefore, we assume that there is no A present in the bulk liquid, so that $C_b = 0$.

$$C_A(0) = C^*$$
$$C_A(\delta) = 0$$

The solution can be written in terms of exponential functions or hyperbolic functions, which are linear combinations of exponential functions. It is convenient to choose the latter form, writing

$$C_A = C_1 \cosh\sqrt{\frac{k}{D}}\,x + C_2 \sinh\sqrt{\frac{k}{D}}\,x$$

Evaluating the constants using the boundary conditions eventually leads to

$$C_A(x) = C^*\left[\cosh m\,X - \coth m \sinh m\,X\right]$$

where $m = \sqrt{\frac{k}{D}}\,\delta$ and $X = \frac{x}{\delta}$.

We can estimate the flux of A at the interface as

$$N_{Ax}(0) = -D\,\frac{dC_A}{dx}(0) = \frac{D\,C^*}{\delta}\,m\coth m$$

Here, we have used the assumption that the solution is dilute so that we can neglect the convective flux that arises from diffusion.

For the same problem in the absence of reaction, we would have obtained

$C_A = C^*(1-X)$ and a flux $N_{Ax}(0) = \dfrac{DC^*}{\delta}$. Therefore, the reaction causes an enhancement of the mass transfer rate. We can define an enhancement factor E as the ratio of the flux of A at the interface in the presence of reaction to the flux of A when there is no reaction. Then, we can write

$$E = m \coth m = m\frac{\cosh m}{\sinh m}$$

Therefore, we can see that the film model is predictive here, suggesting a specific enhancement of mass transfer in the presence of a first order chemical reaction. This can be tested by experiment.

Another Illustration of the Use of the Film Model – High Mass Transfer Rates

As a second illustration, consider the transport of a species A through a mixture of species A and B under conditions where the convective flux arising from diffusion cannot be ignored. Assuming constant physical properties, steady unidirectional transport, and no chemical reactions, leads to the following result once again for the flux of A in the x-direction.

$$\frac{dN_{Ax}}{dx} = 0$$

This means that the flux N_{Ax} is constant along the diffusion path. We can show in a like manner that the flux N_{Bx} will be a constant as well, and therefore the total molar flux N_t also is a constant. Now, we write Fick's law as follows.

$$N_{Ax} = x_A\,(N_{Ax} + N_{Bx}) - C\,D\frac{dx_A}{dx} = x_A\,N_t - C\,D\,\frac{dx_A}{dx}$$

Define

$$\Phi = \frac{N_t}{DC/\delta};\quad \beta = -\frac{N_{Ax}}{DC/\delta};\ \text{and}\ \ \eta = \frac{x}{\delta}.$$

Then, the result from Fick's law can be rewritten as the following differential equation for the mole fraction of A along the diffusion path.

$$\frac{dx_A}{d\eta} - \Phi\,x_A = \beta$$

Using the boundary conditions

$$x_A(0) = x_{A1}; \quad x_A(1) = x_{A2}$$

the solution can be written as follows.

$$\frac{x_A - x_{A1}}{x_{A2} - x_{A1}} = \frac{\exp(\Phi\eta) - 1}{\exp(\Phi) - 1}$$

Note that we now have both a diffusive and a convective contribution to the flux of A. The diffusive flux in this situation is labeled J_A^*. It is given by

$$J_A^* = -C\,D\,\frac{dx_A}{dx}$$

From the solution, we find that at the interface $x=0$,

$$J_A^*(0) = \frac{CD}{\delta}(x_{A1} - x_{A2})\frac{\Phi}{\exp(\Phi) - 1}$$

whereas, in the dilute solution limit where we would have set the convective flux to zero, we would have obtained a diffusive flux given by

$$J_A^*(0) = \frac{CD}{\delta}(x_{A1} - x_{A2})$$

Therefore, the influence of convective transport is to correct the diffusive flux by a multiplicative factor $\dfrac{\Phi}{\exp(\Phi) - 1}$ at the interface. When $N_t > 0$, $\Phi > 0$, and this correction is always less than unity. Therefore, for mass transport into a stream, the convective drift correction reduces the diffusive flux of A. As a corollary, for mass transfer out of the stream, $N_t < 0$, the gradient is strengthened, and the correction is greater than unity.

Of course, the total flux of A is made up of the diffusive and the convective contributions. The total flux of A at the interface is

$$N_A(0) = x_{A1}\,N_t + J_A^* = \frac{CD}{\delta}\left[x_{A1}\,\Phi + (x_{A1} - x_{A2})\frac{\Phi}{\exp(\Phi) - 1}\right]$$

The Penetration Model

In the penetration model of mass transfer, which is used for describing transport in a stirred vessel, we imagine an element of liquid that is present in the bulk being brought by the agitation to the interface with another fluid that can be a gas or liquid. The element spends some amount of time at the interface, and mass transfer occurs between the two phases during this period. Then, the element enters the bulk liquid again and the contents are dispersed. The rate of transfer of a species A from the upper phase to the stirred liquid is estimated by considering a distribution of ages of the elements of liquid at the interface. Crucial to this picture is the idea that a liquid element spends some specified amount of time being exposed to the other fluid. If we presume the concentration of A at the interface to be given by C*, and imagine the depth of penetration of the diffusing species A into the liquid to be small compared with the thickness of the element in the direction of diffusion, it can be shown that the solution $C_A(t,x)$ for unsteady one-dimensional transport, assuming a dilute system and constant physical properties, is given by

$$\frac{C^* - C_A}{C^* - C_b} = erf\left(\frac{x}{2\sqrt{Dt}}\right)$$

where x is the distance from the interface. The diffusive flux of A into the liquid at the interface is given by

$$N_{Ax}(t,0) = -D\frac{\partial C_A}{\partial x}(t,0) = (C^* - C_b)\sqrt{\frac{D}{\pi t}} = k_c\,(C^* - C_b)$$

We can see that the mass transfer coefficient obtained from this flux is proportional to the square root of the diffusivity. The penetration model correctly describes the mass transfer situation where a flowing liquid film is exposed to a gas and a species A is absorbed into the liquid. This problem is discussed in detail in Section 18.5 of the text by Bird et al. Other examples include transport to and from jets, and transport to and from drops and bubbles.

The main features of the film and penetration models can be summarized as follows.

The film model assumes transport to occur in a fictitious film at the phase interface. Outside of this film, concentrations are uniform and assumed to be at the bulk values. The film is assumed to be stationary regardless of the fact that the materials on either side of the interface are usually in motion. Transport in the film occurs only by diffusion and only in the direction normal to the interface. The effect of convective transport in the main flow direction is accommodated through the adjustment of the film thickness to match experimentally obtained mass transfer rates. The film model predicts that mass transfer rates are proportional to the first power of the diffusivity, even though this can be altered by making the film thickness depend on the diffusivity. The film model is commonly used to describe mass transfer in devices such as absorption and distillation columns and catalytic reactors, to cite a few examples.

In the penetration model, the idea is used that fluid elements at the interface are exposed to the fluid on the other side for a finite amount of time and in a reference frame riding with these elements, unsteady diffusion occurs in the direction normal to the interface. It yields predictions for mass transfer rates that are proportional to the square root of the diffusivity. Examples where the penetration model is used include mass transfer from a gas to a liquid in a stirred vessel, mass transfer in a falling liquid film or a jet of fluid, and mass transfer to and from drops and bubbles in absorption and extraction operations.

Reference

R.B. Bird, W.E. Stewart, and E.N. Lightfoot, Transport Phenomena, Wiley, 2007.

Concluding Comments

In these notes, several topics have been omitted. Some examples are

1. Turbulent transport

2. Radiative transport of energy

3. Simultaneous Energy and Mass Transport

4. Complex conduction or diffusion problems

The aim has been to provide a brief introduction to the modeling and analysis of the transport of momentum, energy, and species, identifying the analogies among the three, and to illustrate some general methodology with example problems. There has been no attempt to be comprehensive in the treatment of transport phenomena, as might be suitable for a textbook on the subject.