Fluid dynamics and heat transfer

An introduction to the fundamentals

Brian D. Storey

Olin College

Contents

1
Introduction

The text lays the foundations for the study of fluid dynamics and heat transfer. The first question a student might ask is why should we be interested in fluid dynamics and heat transfer? Let me address the question through some examples of applications where these topics are relevant.

Biology: Our bodies are mostly water. Fluids are essential to life as we know it. The understanding of the transport of oxygen and nutrients throughout the body by fluids (at the level of lungs, heart, and blood vessels as well a the cellular one) requires a basic understanding of the topics herein. Locomotion of many species from fish to birds to small microorganisms is determined by fluid flow.

Geophysical: The atmosphere and oceans are fluids. The currents, the winds, the gulf stream, and the jet stream carry heat and matter around the planet. Our basic understanding of the weather, the climate, and the environment can only begin with a basic understanding of fluid dynamics. The Earth's magnetic field is generated by convection of the mantle deep within the Earth.

Energy: Most of our current energy needs are met by the combustion of fuel; the basic processes of combustion depend upon the details of the fluid mixing of fuel and air and and the resulting heat release. Advances in understanding of the details of these processes has led to a number of developments over the years which made combustion cleaner and significantly reduced the number of pollutants emitted from cars. Many of the renewable sources of energy such as wind, geothermal, ocean waves and tidal seems obvious to state that fluid flow plays a

prominent role. It is fair to claim that an understanding of all energy systems must begin with the basic understanding of topics in this text.

Climate The computer models used by climate scientists begin with the basic equations we will develop here. Since we can only run a single irreversible real-time experiment on our planet's climate, computer models are needed to understand what is likely to happen to the climate in the coming centuries. One cannot appreciate the complexity, accuracy, and shortcomings of these computer models without understanding the basics of fluid transport.

Infrastructure and transportation: Fluid dynamics and heat transfer also play a role in our basic civil infrastructure; water delivery to our homes, water resources management, sewage, building heating and cooling. Losses due to aerodynamic drag on boats, cars, and planes set the efficiency and energy use of most transportation systems.

Engineering: Many fields of engineering need some understanding of flow and heat transfer. Aerospace engineers need to understand fluid flow to get planes to fly. Automotive engineers need to understand the processes of combustion, the lift and drag forces on the vehicle, as well as how to keep the engine cool. Material scientists need to understand heat flow for many thermal processes. Civil engineers need to understand water management in cities. Computer engineers need to understand that dissipation of heat is one of the main limitations in making faster computer chips. As argued before, our bodies are mostly water, thus it seems obvious that bio-engineers need to understand fluids as well.

Astronomy: Despite the extraordinarily large scale, the formation of galaxies, planets, and stars can be studied with the equations of fluid dynamics. We cannot test experimental galaxies in a lab, thus fluid dynamics simulations is an integral part of answering questions about the formation of objects in our universe.

Mathematics: Fluid dynamics has also provided a rich field of study for mathematicians. The basic equations of fluid dynamics, the Navier Stokes equations, are notoriously difficult to solve. If you can provide a general solution, you can win a million dollars (see the Millennium Prize problems sponsored by the Clay Mathematics Institute). There are many cases where in an attempt to solve a problem in fluid dynamics has lead to new and powerful mathematical techniques. Computational fluid dynamics, solving the Navier Stokes equations numerically, has

been one of the most powerful drivers in developing supercomputers and other advances in high performance computing.

In short, while my viewpoint is biased, the foundations in this book are critical to essentially all scientists and engineers.

1.1 Scope of the text

Now that I have built up the number of applications this field is relevant to, we must scale things back. A study this comprehensive would overwhelm most mortals over their lifetime. We only begin our study of these topics and this course will provide only a basic background. This text will mostly provide the theoretical and mathematical foundations. The book is not meant to stand-alone but should be part of course which also involves a balance of working examples, conducting some simple hands-on experiments, and learning how computer packages can help solve some of these problems.

It is expected that this text will provide the (somewhat) rigorous background and derivations. You should read and work through many of these results yourself. Even though the text is not too long, many of the chapters are very dense in terms of content. In my course we will review some of these high points in class, but I will leave it to you to really read through the details. The details can sometimes be tedious, and until you have some working knowledge (perhaps by the end of the course), it is hard to see why one would care about them. These notes should hopefully provide you a view that the mathematical foundations for the field are relatively rigorous. In many cases, we can write the equations down in a few short lines and we could in theory solve any problem. We will soon see that matters are not so simple and that while the equations are known, their solution can be extraordinarily challenging.

Finally, there are a number of good books on these topics. I will never provide some insight that was not thought of before. My treatment will rely on a number of excellent books through which I learned the topic. So why bother to write my own notes? The basic answer is that professors can't help themselves. The way everyone else explains stuff is never quite the way you want it. However, the excellent books that I use the most and those who influenced my view are,

- *Fluid Mechanics* by Kundu and Cohen (2004).
- *Transport Phenomena* by Bird et al. (1960).
- *Div grad curl and all that* by Schey (1997).
- *Fundamentals of Heat and Mass Transfer* by Incroprera and DeWitt (2001).
- *Incompressible flow* by Panton (1996).
- *Album of fluid motion* by VanDyke (1982).

1.2 What is a fluid?

We have already talked about fluids a lot. The two fluids we will discuss most often are air and water. However, one might ask what is a fluid? A precise definition seems difficult. The most popular example of a substance that is not clearly a fluid or a solid is Silly Putty. Put it on the table, leave it, and it flows. Drop it and it bounces. Silly Putty is just one example of a complicated material, though many more exist. Substances which we normally think of as a solid can act like a fluid (and vice versa) under extreme conditions. Hit water hard enough and it can fracture. Wait long enough and a mountain can flow. You should search the internet for "pitch drop experiment" to see an interesting long term experiment. A funnel of pitch tar was placed in a funnel in 1927 and has been dripping ever since. To date nine drops have fallen, so the material does flow. However, hit the material with hammer and it easily shatters. Sometimes the difference between a fluid and solid is just a matter of time scale.

The primary property that differentiates fluid from solid behavior is that fluids cannot support shear forces at equilibrium. If I take a block of solid material between my hands and shear it, the solid resists the motion. The solid may deform, but the solid can resist and ultimately stop the motion. Fluid on the other hand, will just flow and cannot stop the shearing force. I can shear the fluid continuously.

We will be a little more precise later when we develop a mathematical theory. We will see we need to define a constitutive law which relates deformation to forces. The simplest fluid model is called a Newtonian Fluid, which turns out to describe air, water, and oils quite nicely. It does not describe Silly Putty. Later we will define fluids based on their

constitutive laws. For now just go with your intuition - if it flows, its a fluid and not get caught up in a precise definition just yet.

1.3 Simple fluid properties

There are two common fluid properties that we will refer to throughout this book, density and viscosity.

Density which typically has the symbol, ρ, is nothing more than the mass per unit volume. Everything we discuss in the development of the theory is given on a per unit volume basis, thus the density will show up throughout the course. Density of a liquid is really easy to measure, just fill a beaker of known volume and weigh the mass of the fluid. The ratio of mass divided by volume is density. When we write our fluid version of Newton's laws, density will take the role of mass when we write $F = ma$.

Viscosity which typically has the symbol μ (though η is common in chemical engineering) describes a substance's resistance to flowing. You probably have some intuition about this concept based on pouring water and syrup. A simple device for quantifying viscosity comprises two concentric cylinders with a very narrow gap between them. A fluid of interest is placed in the gap. The inner cylinder is held fixed while the outer one rotates at a constant angular speed, ω. The speed of the wall at the outer cylinder is simply $U = \omega R$, where R is the radius of the cylinder. If we conducted this experiment for different gap sizes d and different speeds, we would find the torque needed to spin at constant speed would follow a law,

$$\text{Torque} \propto \mu \frac{U}{d}.$$

We would find for all things equal, syrup is harder to spin than water because the syrup has a higher viscosity. Not all fluids would follow the law above. Only simple fluids such as water, air, syrup, and oil. Fluids that follow a law such as that above are called Newtonian, a definition we will be precise about later.

1.4 Thermal properties

The two common thermal properties of fluids are thermal conductivity and specific heat.

Thermal conductivity which typically has the symbol k tells us how easily the substance conducts heat. Thermal conductivity applies to solids, liquids, and gases. Your intuition about thermal conductivity probably comes from solids. If I placed a copper spoon and a wooden spoon in a pot of boiling water and told you you had to grab one of them - if you picked the wooden spoon then you know about thermal conductivity. If I maintain a the same temperature difference across two materials, more energy will flow through the one with higher thermal conductivity. If you are familiar with the idea of electrical conductivity, thermal conductivity is the same thing (and in fact, materials that conduct electricity well also conduct heat well).

Specific heat which typically has the symbol c and tells us how much thermal energy is stored per unit mass, per degree. You might be familiar with an expression such as

$$\Delta E = mc\Delta T$$

which says that he change in thermal energy of an object (heat) is equal to the product of the objects mass, the temperature changes and the specific heat.

The specific heat is easiest to understand (for me) when discussing a gas. In a gas, temperature is equivalent to the kinetic energy of the molecules. What we sense as macroscopic temperature is the average velocity of the molecules whizzing about. However, for a molecule such as oxygen, O_2, the molecule looks like a dumbbell. The molecule can move about with kinetic energy which we would sense as temperature. However, the molecule can store energy in vibration (the two O atoms can vibrate like two masses on a spring). Energy can also be stored in rotation of the molecule spinning. Energy stored in vibrational and rotational modes *do not* contribute to the temperature. A monatomic gas such as argon is a point mass and cannot store energy in vibration or rotation. Thus in a monatomic gas all the energy is kinetic and thus proportional to temperature. The specific heat is connected to the nature of the molecules. The more complicated the molecule, the more energy can be stored internal modes. Thus the specific heat for

a complicated gas molecule is *greater* than that of a monatomic gas. For the same amount of energy change, the temperature change is less in the complicated molecule because some of the energy is taken up in modes other than kinetic. This picture of specific heat is simplified, but provides a little insight that specific heat is connected to the molecular nature of the material of interest. We will treat c as a constant that has been measured and we look up.

The final confusing thing about the specific heat is that there are really two specific heats, constant pressure specific heat, c_p, and constant volume specific heat, c_v. The notation refers to what is held constant when the measurement is taken, pressure or volume. For solids and liquids there is no significant difference between the constant pressure and constant volume specific heats, so we just use the symbol c. For gases, the difference is very important. We will not get too caught up in the subtleties yet, but it is important to keep the difference in mind.

1.5 Other properties

There are many other properties of fluids that show up but will not play much of a role in our initial study.

Surface tension is a usually given the symbol σ and has units of force per unit length. Surface tension exists at the interface of a liquid with a gas and it is the force that holds raindrops together and makes them want to be spherical. Surface tension can hold relatively small objects like water striders from sinking into the pond. Surface tension can hold water droplets to the ceiling tiles in your shower. Surface tension also acts at the interface between two immiscible liquids like oil and vinegar. If there is no free surface in the liquid then surface tension does not play a role. For flows at a liquid interface for objects the size of boats and people, surface tension is too small to matter much. For objects that are measured in millimeters, surface tension can be a dominant force.

Speed of sound is exactly what it says - it is the speed that sound travels in the fluid. The speed of sound becomes important when flow speeds start to approach it. The ratio of flow speed (or speed of the object moving through the fluid) to sound speed is called the Mach number, Ma. When the Mach number exceeds about 0.3, then the flow

becomes compressible meaning that the density can no longer be considered a constant. When the speed of the object becomes greater than the sound speed, Ma > 1, then we have supersonic flow. Supersonic flow is very interesting as much of our intuition about fluid flow goes out the window. Supersonic flow is a topic for another course.

Elasticity Normally we think of solids exhibiting elasticity. I pull on a rubber band it will stretch. When I let go it will return to its initial state. Simple fluids like water, air, syrup and simple oils do not exhibit any significant elasticity. If I shear the fluid and then let go, the fluid does not try to return to its original configuration. Some fluids such as biological fluids or polymeric solutions exhibit both elasticity and viscosity - typically called viscoelastic fluids. Silly putty is a classic example, I can roll it in a ball and it will bounce or leave it on the table and it will puddle up. In the case of silly putty, it is the time scale of the flow that distinguishes between whether the flow is more viscous or more elastic. Viscoelastic fluids have many interesting properties but will not appear in our initial study.

Other viscosity parameters The basic behavior of simple fluids such as water and air are the only one we will consider. Fluids that obey our basic viscosity law are called Newtonian fluids, thus anything that does not follow this law is generically called Non-Newtonian. The term Non-Newtonian fluid is not very descriptive as it applies to any behavior which is, well, not Newtonian. We will be more precise later about what we mean by a Newtonian fluid.

Beyond viscoelasticity, another non-Newtonian behavior is called shear thickening. Shear thickening means that the faster we stir the fluid, the hard it becomes to stir. Cornstarch mixed with water is an example of this behavior - you can find videos where people run across a pool of it but when they stand still they sink. We can also have shear thinning, meaning that the faster we stir the easier it is to stir. We can also have fluids that act as a solid as long as the applied forces are small, but once we exceed some threshold they start to flow. One could spend a career (and many have) just studying Non-Newtonian fluids.

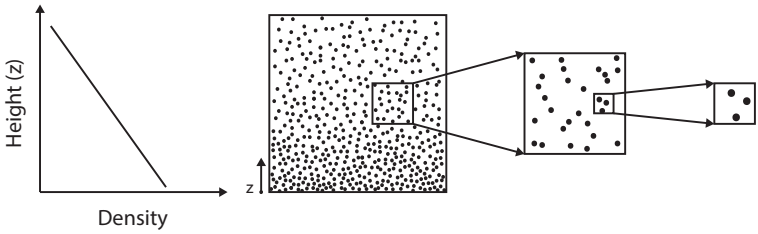


Figure 1.1 Schematic of the concept of a continuum. The local mass density of a gas varies with height in a room with less dense fluid moving to the top. If we zoom in to sample the density we can obtain a local value by using smaller and smaller measuring volumes. At some point, we trapped so few molecules in the volume that the notion of density at such a small scale loses meaning.

1.6 What is a continuum?

In this course we will develop a theory where materials can be considered to be a continuum. We will generally not be concerned that the fluid or solid is made up of molecules or atoms. This assumption works well in applications where the features we are interested in studying are much larger than the distance between molecules. The mean free path of air at standard conditions is approximately 0.1 μm. As long as the smallest features of the problem are greater than this scale, the continuum approximation is a good one. In liquids the mean free path is usually a few angstroms (an angstrom is 10^{-10}m), making cases where we must account for non-continuum effects rare for an engineer.

To be more precise, let's consider a measurement of the density of air as an example. We could measure the density by taking a known volume of air and weighing it. If our volume was the size of the room, the density measure would be the average density in the room. As we shrink our measuring volume, the value we measure for the density might change. The density of warm air is less than cool air, so if we took our measuring volume near the ceiling we might expect a different answer than near the floor. This idea is shown schematically in Figure 1.1, where the local measurement of density is shown to be linear with height, with less dense fluid rising to the top. As we shrink the measuring volume we obtain a more *local* measure of the density. At some

point we keep shrinking the measuring volume and we keep getting the same answer. This constant value would be what we would consider the local, continuum value of the density at that point in space. If we continued to shrink the measuring volume we run into trouble with the fact that air is comprised of many molecules. Eventually the measuring volume would be so small that the mass we measure would depend upon how many molecules we managed to trap. If there are only 10 molecules on average in our volume, then the answer starts to fluctuate depending on if we randomly had 8, 9, 10, 11, or 12 molecules in the volume at the time of measurement. The continuum limit is thus the intermediate asymptotic, where we take the limit such that we can talk about local quantities (i.e. density, temperature, pressure, velocity) at a point in space, but we don't take the limit too small that the molecular nature of matter comes into play. As long as the geometry of the problem is large compared to the molecular scale, then the continuum assumption is good.

1.7 Mathematical prerequisites

I will assume throughout that the reader is familiar with Newton's Laws, Calculus (integration and differentiation), and basic vector calculus. If you know $F = ma$, can compute some simple integrals, and have heard of the divergence, gradient, and curl, you are probably good. If it has been some time since you took vector calculus, I would recommend the book *Div, Grad, Curl and All That* as a complement to these notes. A few common things that we will make use of are provided below.

We will make use of the Fundamental Theorem of Calculus which states

$$\int_a^b \frac{df}{dx} dx = f(b) - f(a).$$

We will use vector calculus notation such as the gradient of a scalar function $F(x, y, z)$ is a vector given as

$$\nabla F = \frac{\partial F}{\partial x}\mathbf{i} + \frac{\partial F}{\partial y}\mathbf{j} + \frac{\partial F}{\partial z}\mathbf{k},$$

where $\mathbf{i}, \mathbf{j}, \mathbf{k}$ are the unit vectors in the x, y, z direction. We will use the

convention that bold face lower case letters are vectors. The divergence of a vector field $\mathbf{f}(x, y, z)$ is a scalar given as

$$\nabla \cdot \mathbf{f} = \frac{\partial f_x}{\partial x} + \frac{\partial f_y}{\partial y} + \frac{\partial f_z}{\partial z}.$$

The primary vector calculus theorem we will use is the divergence theorem, which states,

$$\int \nabla \cdot \mathbf{f} dV = \int \mathbf{f} \cdot \mathbf{n} dS.$$

This theorem equates the flux of a vector field through a closed surface to the volume integral of the divergence of the vector field. Above, we use the notation that dS is a unit of surface and dV is a unit of volume.

If if the above notation seems vaguely familiar to you but you have forgotten the details, you will have no problem. We will review vector calculus in a later chapter. If you have never seen these symbols before, you might have some trouble understanding some of the later chapters.

2

Dimensional analysis

The result of any physical equation must remain unchanged regardless of the system of units that we select for the problem. While this seems like a obvious statement the implications are quite interesting. The simple fact that units are man-made and thus arbitrary can lead to an extraordinary simplification in many problems. The number of free parameters in any physical problem can be reduced by taking the units out of the problem. This concept is not limited to the topics in this book, however it seems that the field of fluid dynamics and heat transfer have adopted this viewpoint to the extent that the discussion of dimensional analysis is in every book. As we will soon show, the reasons for adopting a dimensionless world-view are quite compelling. Books of experimental data can be reduced to a single chart.

However, the dimensionless world-view can be overdone and only working in a dimensionless world can obscure some intuition about the physics of a particular problem. We will try and balance the dimensional world-view and the dimensionless one throughout this book.

2.1 Units

We often refer to fundamental and derived units. Fundamental units in this course will be mass (M), length (L), time (T), and temperature (Θ). Derived units are things like velocity (L/T), force (ML/T^2), or density (M/L^3). The distinction above is arbitrary. Of course we could say that force is fundamental and mass is derived. If you want to do

this, this is your right. I find it easiest to stick with the distinction above.

We could relate temperature and energy and thus remove temperature as a fundamental unit. In this course, this path would not lead to simplification. It would require us to bring in a new parameter, Boltzmann's constant. Boltzmann's constant bridges the statistical definition of kinetic energy of molecules to the macroscopic temperature. If we are not dealing with statistical mechanics, then introducing Boltzmann's constant just leads to confusion. In this course, everything is macroscopic and thus treating the temperature as its own fundamental unit is always going to be the way to go.

2.2 Buckingham Pi Theorem

The Buckingham Pi Theorem is the starting point for any discussion of dimensions. The Pi Theorem states that if a problem has N independent parameters and there are R independent dimensions, then there are N-R independent parameters in the problem. This simple theorem can only be understood through example.

2.2.1 Pendulum

The first example on dimensional analysis every book uses is the pendulum. I won't be original. Imagine a point mass (m) hanging on a massless string of length (ℓ) in a gravitational field (g); shown schematically in Figure 2.1. There is no friction. All this is an idealization, but it is one that we can easily realize in practice to a good degree of approximation. The question is what is the period of oscillation, t, for a given starting angle θ and how does it depend upon the parameters in the problem? There are five independent parameters (m, ℓ, g, θ and t). These parameters are made up of 3 independent dimensions (M, L, and T). Applying the theorem tells us that $N = 5$, $R = 3$ and thus there are only 2 independent parameters.

There are a number different ways to formally approach the removal of units from the problem to decide what the dimensionless parameters are. Once you are comfortable, you can often just remove the units by inspection. I like the table method as it is a provides a good framework

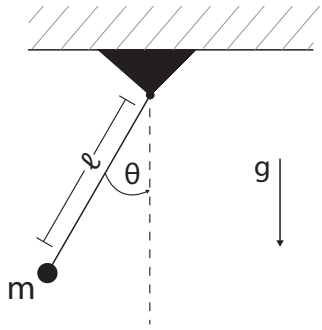


Figure 2.1 Schematic of a simple pendulum.

for formally approaching the problem. We make a table with all the parameters and their units in brackets. We put the primary thing we want to know, in this case the period, in the first column. For this problem, our table starts as,

$t\ [T]$	$m\ [M]$	$\ell\ [L]$	$g\ [\frac{L}{T^2}]$	θ

Note that the starting angle θ has no units. Angles are measured in radians which is defined as the arc length divided by the radius, thus the radian is already dimensionless.

Now start adding lines to the table, and in each new line remove one of the independent dimensions and one dimensional parameter from the problem. Let's start by removing mass from the problem. Since the dimension M only exists in the mass, there is nothing to do but remove it from the problem. *Mass cannot be a parameter to the problem as there is no way to cancel that fundamental unit.* With each new line to the table it is useful to add yourself a little note to the right.

$t\ [T]$	$m\ [M]$	$\ell\ [L]$	$g\ [\frac{L}{T^2}]$	θ	
$t\ [T]$		$\ell\ [L]$	$g\ [\frac{L}{T^2}]$	θ	remove M

Now let's remove length from the problem by defining a new parameter g/ℓ that does not have length in it. We expand our table by one line so it reads,

$t\ [T]$	$m\ [M]$	$\ell\ [L]$	$g\ [\frac{L}{T^2}]$	θ	
$t\ [T]$		$\ell\ [L]$	$g\ [\frac{L}{T^2}]$	θ	remove M
$t\ [T]$			$\frac{g}{\ell}\ [\frac{1}{T^2}]$	θ	remove L

Finally, we can remove time from the problem by adding another line to our table as follows,

$t\ [T]$	$m\ [M]$	$\ell\ [L]$	$g\ [\frac{L}{T^2}]$	θ	
$t\ [T]$		$\ell\ [L]$	$g\ [\frac{L}{T^2}]$	θ	remove M
$t\ [T]$			$\frac{g}{\ell}\ [\frac{1}{T^2}]$	θ	remove L
$t\sqrt{\frac{g}{\ell}}$				θ	remove T

The final result is that there are two parameters, $t\sqrt{g/\ell}$ and θ. We can then state that the answer has the following functional form,

$$t\sqrt{\frac{g}{\ell}} = f(\theta) \quad \text{or} \quad t = \sqrt{\frac{\ell}{g}} f(\theta)$$

At this point we don't know what the function is, but it says the period of any pendulum of any length in any gravitational field only depends upon its starting angle. We only need to measure time in units of $\sqrt{g/\ell}$, and all pendulums are the same. If we always started the pendulum from the same angle, the function on the right hand side would then just be a constant. The result tells if the length of the string is increased by a factor of 4, then the period increases by a factor of 2 to keep $t\sqrt{g/\ell}$=constant. You can easily confirm this result experimentally.

2.2.2 Pythagorean theorem

Dimensional analysis can easily be used to prove the Pythagorean theorem, $a^2 + b^2 = c^2$ for a right triangle. The area of the triangle A only depends upon the angle θ and the length of the hypotenuse c. We don't even need the table, we could move directly to the result that the area divided by c^2 is a unknown function of theta, $A/c^2 = f(\theta)$. The areas of triangles 1 and 2 are thus $A_1/a^2 = f(\theta)$ and $A_2/b^2 = f(\theta)$. Since $A_1 + A_2 = A$, then $a^2 f(\theta) + b^2 f(\theta) = c^2 f(\theta)$. Since the function f must be the same for all triangles, then we can divide it out and obtain the desired result.

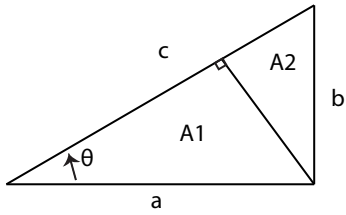


Figure 2.2 Schematic of a right triangle. It is easy to prove $a^2+b^2 = c^2$ using dimensional analysis.

2.2.3 Caveats

There is a nice paper written by Lord Rayleigh on the "Principle of Similitude" (Rayleigh (1915)); note that similitude is another name for dimensional analysis. He starts the paper;

I have often been impressed by the scanty attention paid even by original workers in physics to the great principle of similitude. It happens infrequently that results in the form of "laws" are put forward as novelties on the basis of elaborate experiments, which might have been predicted a priori after a minutes' consideration.

The Rayleigh paper then gives, in essentially 1 sentence each, 13 examples of answers to various problems in physics. The true analytical answer to most of these problems is not trivial. However, the result Rayleigh provides is obtained by essentially applying the Pi Theorem. His results are all correct and some are as simple to obtain as the pendulum example. For many of the examples, if you were given the list of variables which the problem depends on, you could probably reproduce Rayleigh's answers quite readily. However, knowing which parameters matter and which don't is not always a simple task. If you tried to replicate Rayleigh's results from scratch, you would probably have difficulty getting most of the results. You really need to understand the problem before proceeding with dimensional analysis and the Pi Theorem.

There is a textbook on Dimensional Analysis written by P.W. Bridgman in 1922 who makes this exact point (Bridgman (1922)). In Bridgman's book he discusses an exchange that Rayleigh had with another researcher in the journal Nature about Rayleigh's original paper. D. Riabouchinsky questions one of Rayleigh's problem's on heat conduction

arguing that temperature is not an independent dimension since temperature is related to molecular kinetic energy. Rayleigh's reply states that "it would be a paradox if further knowledge of the nature of heat afforded us by molecular theory put us in a worse position than before." Pointing to this full exchange, Bridgman says:

> The reply of Lord Rayleigh is, I think, likely to leave us a little cold. Of course we do not question the ability of Lord Rayleigh to obtain the correct result by the use of dimensional analysis, but must we have the experience and physical intuition of Lord Rayleigh to obtain the correct result also?

For the students, except in the simplest of problems, dimensional analysis via the Pi Theorem, with no additional physical insight and no model from which to extract the right parameters can lead to little insight into the problem and not always a simplification. It usually takes several attempts to understand even a simple problem. As Bridgman says

> The problem cannot be solved by the philosopher in his armchair... only by someone at some time soiling his hands with direct contact.

Thus, when starting to learn dimensional analysis it is important to realize that you will rarely be able to get the right result. You are not Lord Rayleigh. Applying the technique is simple, getting the right parameters is difficult.

As we will soon see, in many problems we know the equations and through a proper formulation of the problem in dimensionless form, we can often extract the right parameters without actually solving the problem. Knowing the equations and being able to formulate a problem mathematically can help us identify the parameters and dimensionless groupings that matter. This approach is the one that will dominate this book.

2.2.4 Drag on a sphere

A simple problem in fluid mechanics is the question, given a sphere of diameter, D, moving at speed, U, through a fluid, what is the drag force, F? It seems reasonable to assume that the flow will depend upon some fluid properties. We haven't talked about fluid properties much, but we introduced the two key properties in the opening chapter. Viscosity has the symbol μ and is expressed in units of $M/(LT)$. Clearly it will be

harder for the sphere to move through the more viscous fluid all other things being equal. It also seems reasonable to suspect that the fluids density, ρ, might matter as well. These are the only two fluid properties that really matter in this problem. We are starting to already see the comments of Bridgman come in. We have to know something about the physics of the problem to know that these are the only fluid properties that matter. Why doesn't the speed of sound in the fluid matter? Or its surface tension? In later chapters as we develop the full theory of fluid flow, we will see that these are the only two properties that emerge for this problem. For now you will have to accept what I am suggesting.

Applying the Pi Theorem helps our problem immensely. There are 5 parameters, F, D, U, μ, and ρ. There are the 3 dimensions of M, L, and T. The Pi Theorem says there are only two parameters that matter. We can proceed by constructing the table and writing all the parameters and their units.

$F\ [\frac{ML}{T^2}]$	$D\ [L]$	$U\ [\frac{L}{T}]$	$\rho\ [\frac{M}{L^3}]$	μ	$[\frac{M}{LT}]$

Immediately we see some arbitrary choices to make. If we try to eliminate mass, we can divide the drag force by the density or the viscosity. According to the Pi Theorem, any choice is just as good. Which choice is "better" is a matter of experience and even an experienced person might try several combinations before deciding which choice they prefer. We will proceed to apply the theorem making arbitrary choices about how to remove the dimensions. (Of course the choices are not arbitrary, they are ones that I selected.) Here is one possibility in the table below. Note that with each line of the table I remove one parameter and one of the independent dimensions.

$F\ [\frac{ML}{T^2}]$	$D\ [L]$	$U\ [\frac{L}{T}]$	$\rho\ [\frac{M}{L^3}]$	μ	$[\frac{M}{LT}]$	
$\frac{F}{\rho}\ [\frac{L^4}{T^2}]$	$D\ [L]$	$U\ [\frac{L}{T}]$		$\frac{\mu}{\rho}$	$[\frac{L^2}{T}]$	Remove M
$\frac{F}{\rho U^2}\ [L^2]$	$D\ [L]$	$\frac{\rho U}{\mu}\ [\frac{1}{L}]$				Remove T
$\frac{F}{\rho U^2 D^2}$		$\frac{\rho U D}{\mu}$				Remove L

The result says there are two parameters. One parameter is ubiquitous in fluid dynamics and we will discuss it extensively in this course. It is called the Reynolds number, $\mathrm{Re} = \frac{\rho U D}{\mu}$. If you have the chance to make a grouping that looks like (density·velocity·length)/viscosity,

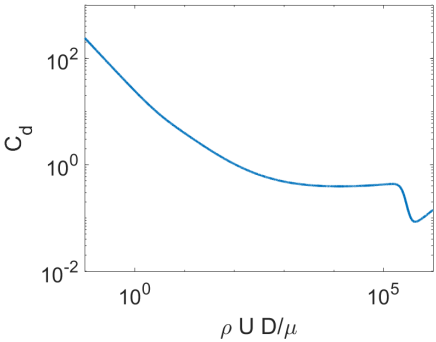


Figure 2.3 Drag coefficient as a function of Reynolds number for flow around a sphere.

you should do so. The other parameter is the dimensionless drag force, known as the drag coefficient. Typically, the drag coefficient is defined by convention to be $C_d = \frac{F}{\frac{1}{2}\rho U^2 \pi R^2}$; which is the same as we got from the table only with a few different arbitrary constants; namely a factor of $1/2$ and πR^2 instead of D^2. There are good reasons that we will come to later in the course for using the constants in the classic form of the drag coefficient. The dimensional analysis result is powerful as the result says there is a single master curve, namely

$$C_d = f(\mathrm{Re}).$$

This single master curve, the function f, captures the drag behavior of all spheres in all fluids. It turns out the details of this curve can be quite complicated and very difficult to calculate from the basic equations of fluid dynamics. However, we can conduct experiments. The dimensional analysis says that if we conduct the experiment of measuring drag as a function of speed once - for one size sphere in a wind tunnel - we now know the result for any other sphere in any fluid. A fit to experimental data on a logarithmic scale is shown as the solid curve in Figure 3.1.

I want to emphasize that dimensional analysis let's us represent data that depends on multiple parameters in a simpler way. In dimensional terms the drag force is given as $F(U, D, \mu, \rho)$. Parameterizing this function experimentally for all four variables requires a lot of experiments. Lets say we would want 10 experimental data points to establish a

reasonable empirical functional relationship of one variable. A single curve on a graph could display force as a function of velocity for a given diameter and given fluid. We could vary sphere diameter to show a family of 10 curves. One graph on one page could summarize 100 experiments. But now we need to vary viscosity. We are now up to 1000 experiments and 10 pages of graphs. Varying the density we need 10,000 experiments and 100 pages (a book). In dimensionless terms, there is a universal master curve of one parameter and that curve is good for all drag experiments with a sphere that have ever been done and all that will be done in the future. The master curve works for all fluids and spheres of any size. We will elaborate on this example in the next chapter.

2.2.5 Pipe flow

Another classic example of a practical use of dimensional analysis is the problem of flow in a pipe. A simple question to ask is given a length of pipe ℓ, with a constant diameter D, with a pressure difference ΔP applied across it, what is the mean flow velocity U? Here, we already have four parameters. Clearly the fluid matters as well so as with the previous problem we will assume that the fluids density and viscosity matter as well.

Before the theory of fluid dynamics was firmly established and the power of dimensional analysis understood, a problem emerged from this seemingly simple example. It is clear that understanding the pressure-flow relationship in a pipe is of great practical importance. While it seems straight forward to establish an experiment to measure flow as a function of applied pressure, there are a lot of parameters to vary in order to obtain a complete model experimentally. Before dimensional analysis was really understood and used, this extensive experimentation is precisely what was done to describe this simple pipe flow problem. There used to be page after page of numerous graphs showing pressure-flow relationships for different pipes and different fluids.

Applying the Pi Theorem helps our problem. There are 6 parameters, ℓ, D, ΔP, U, μ, and ρ. There are the 3 dimensions of M, L, and T. The Pi Theorem says there are only three parameters that matter. This helps quite a bit. We can proceed by constructing the table and writing all the parameters and their units.

$U\ [\frac{L}{T}]$	$\ell\ [L]$	$D\ [L]$	$\Delta P\ [\frac{M}{LT^2}]$	$\rho\ [\frac{M}{L^3}]$	$\mu\ \ [\frac{M}{LT}]$	
$U\ [\frac{L}{T}]$	$\ell\ [L]$	$D\ [L]$	$\frac{\Delta P}{\rho}\ [\frac{L^2}{T^2}]$	$\frac{\rho}{\mu}\ [\frac{T}{L^2}]$		Remove M
$\frac{\rho U}{\mu}\ [\frac{1}{L}]$	$\ell\ [L]$	$D\ [L]$	$\frac{\Delta P}{\rho^2 U^2}$			Remove T
$\frac{\rho U D}{\mu}$	$\frac{\ell}{D}$		$\frac{\Delta P}{\rho U^2}$			Remove L

This result is one of many possibilities. My result says there are three parameters. The first is again the Reynolds number - it will show up in nearly every fluid mechanics problem. The second parameter is a geometric parameter, the length to diameter ratio. The final parameter is the dimensionless pressure drop. Now, conducting experiments for three parameters is a lot easier than 6. We could fit all the results on a single graph plotting the dimensionless pressure drop versus Reynolds number for a family of curves for different ℓ/D. We need a sheet of paper rather than a book to display *all* results.

We can actually do even better, but the comment by Bridgman again applies. We need a little physical reasoning and some knowledge of the problem. This step might seem a leap now, however, as we learn more throughout this course we will make this step more easily. The pressure drop and pipe length are not really independent. It is the parameter $\Delta P/\ell$ that matters most, the pressure gradient. The assumption we make here is that when the pipe is long that any section of pipe is basically the same. If we look at the flow here or there in the pipe, it is just a length of pipe and the flow doesn't change. If the character of the flow doesn't change as a function of length, then doubling the length should not change anything if we double the pressure drop. The pressure drop per unit length is the parameter that really matters. We could confirm this assumption experimentally by measuring the pressure as a function of length and we would find that it is essentially linear with distance. This assumption that pressure gradient matters really only applies to pipes that are long relative to their diameters.

If we use the pressure gradient as our parameter our result is sim-

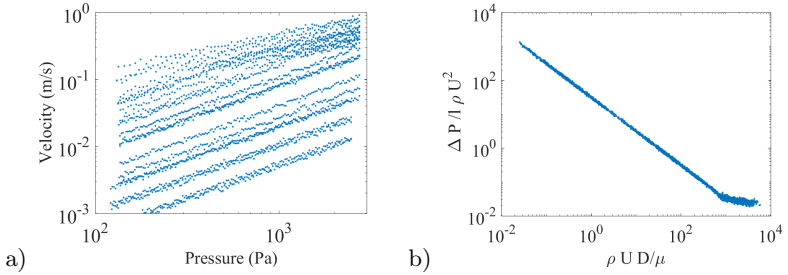


Figure 2.4 Dimensional (a) and dimensionless (b) plots of the flow-pressure drop data for the pipe flow problem. This data was taken in a simple apparatus and consisted of 21 separate configurations varying fluids (water and oil), pipe diameter, and pipe length. The experimental dimensional data in (a) is collapsed to a single master curve in (b). I took this data with a simple table-top device thus the data at high Reynolds number show some significant scatter that can be reduced with some care.

plified as the number of variables is reduced to 5 and the Pi Theorem states we will only have two parameters that matter. Proceeding in the same way to remove dimensions, we could obtain.

$U\ [\frac{L}{T}]$	$D\ [L]$	$\frac{\Delta P}{\ell}\ [\frac{M}{L^2T^2}]$	$\rho\ [\frac{M}{L^3}]$	$\mu\ [\frac{M}{LT}]$	
$U\ [\frac{L}{T}]$	$D\ [L]$	$\frac{\Delta P}{\ell\rho}\ [\frac{L}{T^2}]$	$\frac{\rho}{\mu}\ [\frac{T}{L^2}]$		Remove M
$\frac{\rho U}{\mu}\ [\frac{1}{L}]$	$D\ [L]$	$\frac{\Delta P}{\ell\rho U^2}\ [\frac{1}{L}]$			Remove T
$\frac{\rho UD}{\mu}$		$\frac{\Delta PD}{\ell\rho U^2}$			Remove L

This result is powerful as this is a single master curve. This master curve captures the pressure flow relationship for all pipe sizes, all fluids, and all pressure drops. This is a very powerful result. The punchline of the work is shown in Figure 2.4 where we show the dimensional and dimensionless pressure-flow relationship for pipe flow. I took this data myself in a simple apparatus. There are 21 different experiments in these plots. The pipe diameter and length were varied and data was taken with both water and oil to vary the fluid properties. In dimensional form, the data spans 3 orders of magnitude in velocity. In dimensionless form, all the data collapse to a single master curve. All 21 experiments are described by one curve.

An interesting feature of the data is the kink in the slope. This feature

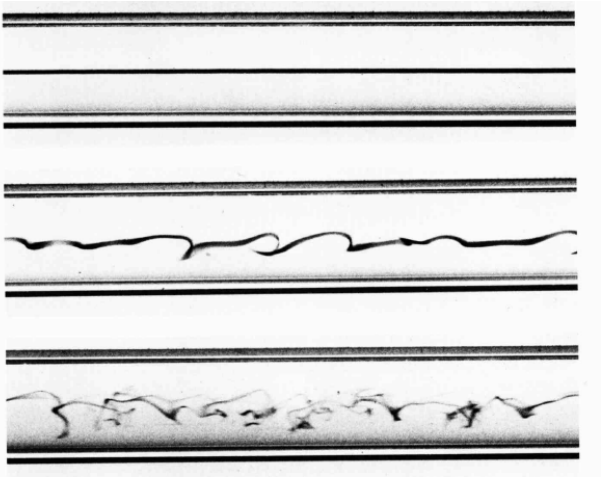


Figure 2.5 Experimental images on pipe flow at different Reynolds number. In the upper image the flow is laminar and injected dye just goes right down the center in a straight line. At higher Reynolds number the flow begins to fluctuate and become unstable. These images were taken in Orbourne Reynold's original experimental apparatus but a century later. In Reynold's original paper he included hand sketches of the flow field. From Album of Fluid Motion.

of the curve was first described by Osbourne Reynolds in one of the most well-known papers in all of fluid dynamics (Reynolds (1883)). At low flow and low pressure, the relationship between pressure and flow is linear. This linear regime is analogous to Ohms law for a resistor, $V = iR$, where voltage drop is replaced by pressure drop and current is replaced by flow. At higher flow rates (pressures) something happens. We will discuss this transition in more detail later, but it represents a transition from a smooth laminar flow to a turbulent one.

Visualization of this instability is shown in Figure 2.5 at three Reynolds number from low to high. In the laminar flow case, the relationship between pressure and flow is easily described by a theory which we will discuss later. In the turbulent case, at high flow, there is no theory that can predict the behavior and even today's fastest computers are limited in calculating turbulent flows. Because the turbulent case is difficult to describe theoretically, the collapse of the experimental data is important. It means from a limited set of experiments we could predict

turbulent flow in any pipe with any fluid. It is also important to realize that he images seen in Figure 2.5 only depend upon the Reynolds number. If we repeated the experiment in water, air, and oil - but we changed the size of the pipe to keep the Reynolds number the same across the three experiments - we would make the same visualizations.

2.3 Making equations dimensionless

In many cases (in fact, most of them in this course), we can actually write down in reasonably compact form the mathematical description of the problem of interest. As we will learn, there are often many difficulties with mathematically solving the problem. In some cases it is hard, in others it is impossible. Recall in the introduction that if you could solve the basic equations of fluid dynamics, the Navier-Stokes equations, in general form you would win a million dollars.

However, in cases where we know the equations but cannot solve them, making the model equations dimensionless can help guide us to pick the "right" dimensionless groups. This equation based approach can be equivalent to working the units through the table approach but is a little bit different in methodology. The table approach will always lead to the same result as the working through the equations. When we have the equations at hand, we can "see" the parameters that matter and will often see where groupings of parameters only appear together in certain ways, which is something that the table approach cannot tell us. Working through the equations can remove a little of the guess work as to what parameters should matter. However, working with the equations does not fully absolve us from the responsibility of knowing the physics of the problem - how do we know what equations are appropriate?

The typical procedure is to define a new set of variables that have no units by scaling the usual dimensional variables by a constant. This sounds crazy but is no different than changing units. In an equation if we have a variable time, t, we could express that time in seconds, minutes, hours, or years. Similarly we can define a dimensionless time, $\tilde{t} = t/t_0$, where both t and t_0 are measured in seconds. The constant t_0 is just some constant that we pick to measure our unit of time in. There is nothing special about the second or the minute, so we can change the

unit to whatever we feel like. Picking the "right" scale for t_0 can be a bit of an art form that comes with experience. However, the equations will serve as out guide us to a "good" answer. Mathematically, t_0 could be anything.

Lets do a few examples to make this clear.

2.3.1 Pendulum

Since we started the chapter with the pendulum example, let's repeat this example using the equation approach. Newton's law, $F = ma$, for a simple pendulum with no friction is,

$$m\ell\frac{d^2\theta}{dt^2} = mg\ \sin(\theta),$$

which is easily rewritten as

$$\frac{d^2\theta}{dt^2} = \frac{g}{\ell}\sin(\theta).$$

We have already recovered the basic result of the Pi Theorem. We have shown that the mass doesn't matter and that the only parameter that matters is g/ℓ. However, the equation still has units so let's proceed to remove them all. The angle, θ, has no units so we leave it alone. Time, t, has units so lets make the transformation $\tilde{t} = t/t_0$ (or equivalently $t = \tilde{t}t_0$).

$$\frac{d^2\theta}{d(\tilde{t}\ t0)^2} = \frac{g}{\ell}\sin(\theta).$$

Note how we made the derivative dimensionless. This step seems strange to most students the first time they see it but we have to remember that a second derivative in time has units of $1/\text{seconds}^2$. Thus, the dt^2 in the derivative is replaced by $d(\tilde{t}\ t0)^2$. All we did was the substitution $t = \tilde{t}\ t_0$. Since t_0 is a constant it can be moved in or out of the derivative operator. This step is no different than changing the time derivative to be measured in minutes instead of seconds.

Our equation can be rewritten as

$$\frac{d^2\theta}{d\tilde{t}^2} = \frac{gt_0^2}{l}\sin(\theta).$$

which is a dimensionless equation with one parameter. However, we have left t_0 unspecified. I am allowed to make it anything I want. It

seems convenient then to define $t_0^2 = \ell/g$ and our equation then becomes,

$$\frac{d^2\theta}{d\tilde{t}^2} = \sin(\theta). \tag{2.1}$$

This final equation has no units and no free parameters.

The solution to the equation requires two initial conditions for the position and the velocity. Expressed mathematically our initial conditions are

$$\theta(t=0) = \theta_0 \quad \text{and} \quad \left.\frac{d\theta}{dt}\right|_{t=0} = \frac{V}{l}$$

where V is the velocity in units of L/T. We now must make these initial conditions dimensionless in the same way as the equations. Under the change of variables we obtain,

$$\theta(t=0) = \theta_0 \quad \text{and} \quad \left.\frac{d\theta}{d\tilde{t}}\right|_{\tilde{t}=0} = \frac{V}{\sqrt{gl}} \tag{2.2}$$

Thus our general problem comprises Equation 2.1 and the initial condition 2.2. The final equation has no parameters and there are two parameters that enter into the initial condition. All pendulums (started from rest and from the same angle) are the same when time is plotted in units of $\sqrt{\ell/g}$ instead of seconds. The scaling for time is exactly as we obtained from the Pi Theorem. We have no new information from this approach than we got from the table.

2.3.2 Mass-spring-damper

As an example where we have additional parameters, consider a simple mass-spring-damper system,

$$m\frac{d^2x}{dt^2} = -\beta\frac{dx}{dt} - kx$$

with the initial conditions that $x(t=0) = x_0$ and the velocity initially is zero.

We will make the equations dimensionless by defining $\tilde{x} = x/x_0$ and $\tilde{t} = t/t_0$. Here, t_0 is arbitrary, but x_0 is the initial position. Using the initial position seems like a good choice to make x dimensionless

because the solution for x will always be bounded in magnitude by 1. Applying the change of variables we obtain

$$m\frac{d^2(\tilde{x}\ x_0)}{d(\tilde{t}\ t_0)^2} = -\beta\frac{d(\tilde{x}\ x_0)}{d(\tilde{t}\ t_0)} - kx_0\tilde{x}$$

Be careful to note the units of d^2x/dt^2. Its units are length/time^2. Thus t_0 is squared whereas x_0 is not. The placement of the squares when we write d^2x/dt^2 are literal; t is squared and x is not. Since x_0 appears in each term it cancels out and we can group t_0 together to obtain,

$$\frac{d^2\tilde{x}}{d\tilde{t}^2} = -\frac{\beta t_0}{m}\frac{d\tilde{x}}{d\tilde{t}} - \frac{kt_0^2}{m}\tilde{x}.$$

Since t_0 is arbitrary we select $t_0^2 = m/k$ to obtain,

$$\frac{d^2\tilde{x}}{d\tilde{t}^2} = -\frac{\beta}{\sqrt{mk}}\frac{d\tilde{x}}{d\tilde{t}} - \tilde{x}. \tag{2.3}$$

with the initial conditions that $x(t = 0) = 1$ and $dx/dt = 0$. Thus, the all mass-spring-damper problems reduce to an equation and initial condition with one free parameter, the dimensionless damping constant. In the absence of damping, all mass-spring systems are reduced to a single problem.

Let's compare our result to what we would obtain by the Pi Theorem, proceeding to remove units using our table method.

$x\ [L]$	$t\ [T]$	$x_0\ [L]$	$k\ [\frac{M}{T^2}]$	$\beta\ [\frac{M}{T}]$	m	$[M]$	
$x\ [L]$	$t\ [T]$	$x_0\ [L]$	$\frac{k}{m}\ [\frac{1}{T^2}]$	$\frac{\beta}{m}\ [\frac{1}{T}]$			Remove M
$\frac{x}{x_0}$	$t\ [T]$		$\frac{k}{m}\ [\frac{1}{T^2}]$	$\frac{\beta}{m}\ [\frac{1}{T}]$			Remove L
$\frac{x}{x_0}$	$t\sqrt{\frac{k}{m}}$			$\frac{\beta}{\sqrt{km}}$			Remove T

The result is consistent with what we obtained from making the equations dimensionless. We can write the result from the table as

$$\frac{x}{x_0} = f\left(t\sqrt{\frac{k}{m}}, \frac{\beta}{\sqrt{km}}\right).$$

2.3.3 Derivatives have units too

An important point that came out in the previous two examples is that derivatives have units too. Whenever we define a dimensionless time

variable based on some "natural" time scale such as $\tilde{t} = t/t_0$, then we need to remember that the dimensionless time derivative will become,

$$\frac{\partial}{\partial t} = \frac{1}{t_o}\frac{\partial}{\partial \tilde{t}}.$$

Whenever we define a dimensionless space variable based on some geometric length scale such as $\tilde{x} = x/L$, then we need to remember that the dimensionless spatial derivatives will become,

$$\frac{\partial}{\partial x} = \frac{1}{L}\frac{\partial}{\partial \tilde{x}},$$

or

$$\frac{\partial^2}{\partial x^2} = \frac{1}{L^2}\frac{\partial^2}{\partial \tilde{x}^2}.$$

It is also important to remember to apply this result to our vector operators which will begin to appear soon. We will work with these ideas in context in the coming chapters, but I want to highlight it now so you don't forget.

2.4 Summary

Throughout this book we will continue to return to the idea of working in dimensionless terms. We will periodically appeal to both the table and equation based approach - realizing that we can get the same results from both approaches. We will see many examples where the mathematical description of a problem can be stated, but it is too difficult (or impossible) to solve the problem. In these cases we can still obtain insight to the problem by appealing to dimensional analysis.

3

Dimensional analysis and the empirical approach

In the last chapter we found that by using physical insight and knowing the important parameters in a problem we could use dimensional analysis to dramatically reduce the number of parameters needed to characterize a problem. We saw through the example of drag on a sphere and flow in a pipe that the complexity of a problem in terms of how many experiments need to be done could be dramatically reduced. All the behavior is described on a single plot or with a single number.

After this current chapter, we will spend much of the rest of the book developing the underlying theory for fluid flow and heat transfer. We will find that this theory can be extremely accurate when compared to experiments and has great predictive power. We will also find that in many cases of practical interest even with great computational resources and sophisticated mathematical tools, many problems are just too difficult to solve. Fortunately, many of these cases are pretty easy to measure experimentally. We often find that dimensional analysis coupled with experimental data can lead to practical solutions and accurate estimates to real world problems. Many times there is no real need for a time consuming or computationally expensive solution.

The focus of this chapter will be to show how dimensional analysis plus experimental data can be put to practical use in an engineering setting. We can think of this approach as purely empirical, While we will only go through a few examples, this approach in a sense could be the end of our practical study. We might not have a lot of physical insight or deep understanding - but we could perform simple calculations, make predictions, and design useful things.

While all the interesting stuff (to me, anyway) will come up in later

chapters, this chapter may be the most useful when you just need to solve problems. We will revisit two examples from the last chapter on drag and pipe flow, and then discuss convective heat transfer. The chapter here is meant to be illustrative and not comprehensive. Many textbooks on engineering fluid dynamics and heat transfer have numerous details and practical situations that have been studied. My aim here is to show this empirical approach first, and then dig into the details a little later in the book.

3.1 Drag revisited

In the last chapter we described a simple problem (well, at least simple to state); given a sphere of diameter, D, moving at speed, U, through a fluid, what is the drag force, F? We stated that the behavior will depend upon two fluid properties - density and viscosity. There are thus 5 parameters, F, D, U, μ, and ρ expressed in the 3 independent dimensions of M, L, and T and thus the Pi Theorem says there are only two dimensionless parameters. The table with all the parameters and their units was found to be

$F\ [\frac{ML}{T^2}]$	$D\ [L]$	$U\ [\frac{L}{T}]$	$\rho\ [\frac{M}{L^3}]$	$\mu\ \ [\frac{M}{LT}]$	
$\frac{F}{\rho}\ [\frac{L^4}{T^2}]$	$D\ [L]$	$U\ [\frac{L}{T}]$		$\frac{\mu}{\rho}\ \ [\frac{L^2}{T}]$	Remove M
$\frac{F}{\rho U^2}\ [L^2]$	$D\ [L]$	$\frac{\rho U}{\mu}\ [\frac{1}{L}]$			Remove T
$\frac{F}{\rho U^2 D^2}$		$\frac{\rho U D}{\mu}$			Remove L

Defining the drag coefficient as $C_d = \frac{F}{\frac{1}{2}\rho U^2 \pi R^2}$ and the Reynolds number as $Re = \rho U D/\mu$, dimensional analysis says there is a single master curve, namely

$$C_d = f(\mathrm{Re}).$$

This single master curve, the function f, is shown again in 3.1.

What is interesting is that we can find two simple limits that work over a wide range of Reynolds number in this data. At values of Reynolds number less than 1, the data is fit by a function $C_d = \frac{24}{\mathrm{Re}}$ and for the range of $10^3 < \mathrm{Re} < 10^5$ the data is fit by a constant $C_d \approx 0.4$. Between $1 < \mathrm{Re} < 10^3$ we see the data smoothly transitions from one limit to the other. Above $\mathrm{Re} > 10^5$ a strange dip in the drag coefficient happens

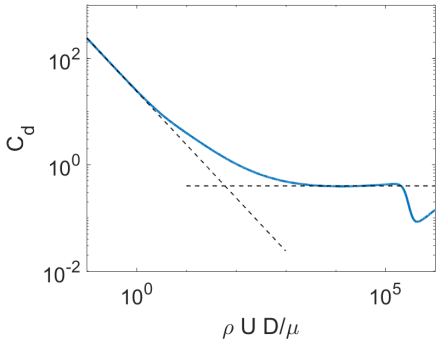


Figure 3.1 Drag coefficient as a function of Reynolds number for flow around a sphere is shown as the solid curve. The dashed lines are for the fit $C_d = 24/\mathrm{Re}$ and a constant of $C_d = 0.4$.

that we will discuss later. We can actually derive the two limits using dimensional analysis. Low values of Reynolds numbers occur for small objects, moving slowly, through a very viscous medium (i.e. look how the terms D, U, and μ show up). These are flows that are dominated by viscosity. The fact that the fluid has mass or inertia turns out to be irrelevant. It takes much more effort to drag the fluid along at low Reynolds number than it takes to push fluid out of the way. If fluid inertia is irrelevant to the physics, then we can remove density from our list of parameters and rework the dimensional analysis.

F $[\frac{ML}{T^2}]$	D $[L]$	U $[\frac{L}{T}]$	μ $[\frac{M}{LT}]$	
$\frac{F}{\mu}$ $[\frac{L^2}{T}]$	D $[L]$	U $[\frac{L}{T}]$		Remove M
$\frac{F}{\mu U}$ $[L]$	D $[L]$			Remove T
$\frac{F}{\mu U D}$				Remove L

Our result now says there is one dimensionless parameter, therefore it doesn't depend on anything and must be a constant,

$$\frac{F}{\mu U D} = \text{Constant}.$$

Recasting in terms of the drag coefficient we can multiply both sides

of the equation by the inverse Reynolds number,

$$\frac{F}{\mu U D}\frac{\mu}{\rho U D} = \frac{F}{\rho U^2 D^2} = \text{Constant}\frac{\mu}{\rho U D}$$

which states that the drag coefficient will be equal to a constant divided by the Reynolds number. The data supports the function of $C_d = \frac{24}{Re}$. The factor of 24 comes from a full calculation of the laws of fluid flow and is also validated by experiments. Constants simply *cannot* come from dimensional analysis.

The result I just "derived" probably seems suspicious to you. If it doesn't, it should. Why am I allowed to throw out density from my list of parameters? My derivation again relies on this comment by Bridgman. You have to understand the physics of the problem to successfully apply dimensional analysis. Since this is just the start of your study, you should have no reason to accept that density is unimportant at low Reynolds number. Remember that I have studied fluid mechanics since graduate school, have read about this problem in numerous books, and have run the experiment and plotted data myself. This problem is easy for me at this point; however, faced with a new unfamiliar problem I have to work hard to get the right answer. With a new problem, I would also not typically believe an answer until is was backed up by a more thorough analysis or experimental data. It will take some time until we study fluid motion in more detail for this example to make physical sense to you. However, the excellent fit to the experimental data at low Reynolds number should convince you that we are on to something. If we continued the graph in Figure 3.1 to lower and lower Reynolds number we would see the fit to $C_d = \frac{24}{Re}$ continue.

The other limit at higher values of Reynolds number where the drag coefficient is approximately constant can also be "derived" from dimensional analysis. In this limit the fact that the fluid has viscosity matters less than the fact that the fluid has inertia. At high speeds we have to move the fluid, which has mass, out of the way for the ball to go by. This explanation is actually a little too simplistic and the truth is more subtle, but let's just assume for now that the viscosity doesn't matter and see what the analysis says.

$F\ [\frac{ML}{T^2}]$	$D\ [L]$	$U\ [\frac{L}{T}]$	$\rho\ \ [\frac{M}{L^3}]$	
$\frac{F}{\rho}\ [\frac{L^4}{T^2}]$	$D\ [L]$	$U\ [\frac{L}{T}]$		Remove M
$\frac{F}{\rho U^2}\ [L^2]$	$D\ [L]$			Remove T
$\frac{F}{\rho U^2 D^2}$				Remove L

Our result now says there is one dimensionless parameter and it must be a constant,

$$\frac{F}{\rho U^2 D^2} = \text{Constant}.$$

We see the above equation is the definition of drag coefficient. The analysis says that drag coefficient is constant if the viscosity of the fluid is irrelevant. Again, I should not have convinced you that it is acceptable to ignore the fluid's viscosity, however the data between a Reynolds number of 10^3 and 10^5 supports this view. If we look at the data we see a dip in the drag coefficient that we will later explain - but it turns out the dip occurs because the fluid has viscosity.

In dimensional terms the drag force is given as $F(U, D, \mu, \rho)$. Parameterizing this function experimentally for all four variables requires a lot of experiments. In dimensionless terms, there is a universal master curve of one parameter and that curve is good for all drag experiments with a sphere that have ever been done and all that will be done in the future. The master curve works for all fluids and spheres of any size. If we understand something about the physics of our problem, we can often find simplifying limits that can be good over a wide range of the parameter space.

That the behavior changes dramatically with Reynolds number can be seen through the flow visualization images in Figure 3.2. At low Reynolds number (here Re=1.5 or lower) the flow looks very symmetric from left to right. In this regime the flow is dominated be viscosity. At Re=26, we see asymmetry and a recirculating wake forming behind the cylinder. This is the effect of inertia - the fluid approaching the cylinder has momentum that carries it to tend to "overshoot" on the backside of the cylinder. As the Reynolds number increases to 2000 and 10,000 the wake becomes very complex and has no resemblance to the flow at low Reynolds number. It is also interesting to realize that the flow field is only dependent on Reynolds number. The experiments in Figure

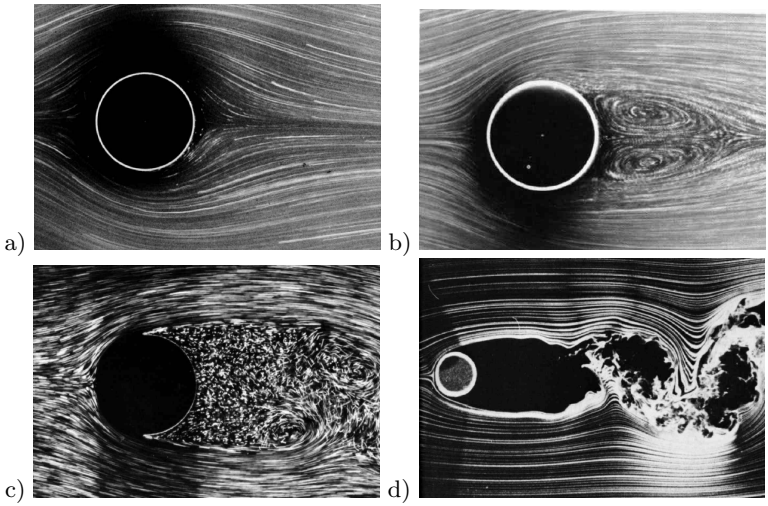


Figure 3.2 Flow around a cylinder at different Reynolds numbers (From Album of Fluid Motion). The Reynolds number is 1.5, 26, 2000, and 10,000 going from figures a to d.

3.2 could be repeated in water, air, or oil and as long as the Reynolds number was matched the flow would look identical.

3.2 Drag coefficients

In summary we have the following two simple formulas for the drag force on a sphere. If the flow is dominated by viscosity then dimensional analysis gave a force on a sphere to have a form

$$F = C\mu U D,$$

where C is constant. If the flow was dominated by inertia the force had a form

$$F = C\rho U^2 D^2.$$

In both cases, the constant C (or Drag Coefficient) is unknown until one conducts additional analysis, numerical simulations, or experiments. Fortunately, for many common shapes drag coefficients have

been measured and can be found in books or the internet. When using drag coefficient data you must take care in noting the form of the formula, and what area or length scale is used.

Most (but not all) drag formulas are written assuming inertia dominated flow. The reason is that most "human scale" problems are in the inertia dominated regime. The transition between the inertia dominated flows and viscosity dominated flows is determined by the Reynolds number, defined as

$$\mathrm{Re} = \frac{\rho U D}{\mu}.$$

The Reynolds number is usually large for things that you interact with on a daily basis - a 1 mm object traveling at 1 mm/s in water will have a Reynolds number of 1. What counts as high or low Reynolds number depends on the situation, but usually greater than 1000 would be considered well in the high Reynolds number regime. The exact number really depends on the situation.

The most common way that the drag coefficient is characterized is,

$$F = C_d \frac{1}{2} \rho U^2 A,$$

where A is the area projected by the object to the flow. It is important to note that this choice of area used in the formula is arbitrary. The area used *must* be reported along with the drag coefficient to avoid any ambiguity. If the drag coefficient is measured but the area that was used to compute C_s is not reported, then the result is useless.

For many objects, the drag coefficient would be reported as a constant. In some cases it will be given as a chart as a function of Reynolds number. If the number is given as a constant, then most likely it is assumed that the Reynolds number is large. Note that if it is given as a constant in a table, it doesn't mean that it is truly constant with Reynolds number as seen in the sphere example. However, looking at Figure 3.1, note how the assumption of constant drag coefficient is pretty good over a wide range of conditions.

A simple search will find a lot of resources and tables of drag coefficients. You will notice that most are around 1 for blunt objects (sometimes around 2, but never 10) and as low as 0.04 for relatively streamlined shapes. We will dig into the physics of this problem again

in a later chapter. For now, if you need to figure out the drag force on an object it is pretty simple (at least conceptually),

- Compute (or estimate) the Reynolds number.
- Search for drag coefficients on the shape of interest.
- Understand what area was used on the reported drag coefficient. Without the area defined, the coefficient is useless.
- If the coefficient is given as a function of Reynolds number, you can just look up on the plot your drag coefficient.
- If a single number for drag coefficients is reported, it is most common for that number to assume high Reynolds number. Be careful that you are using the right Reynolds number range.

3.3 Pipe flow

Another useful example of dimensional analysis is the problem of flow in a pipe which we introduced in the last chapter. Given a length of pipe ℓ, with a constant diameter D, with a pressure difference ΔP applied across it, what is the mean flow velocity U? This problem was discussed previously and the dimensionless table is repeated below.

$U\ [\frac{L}{T}]$	$D\ [L]$	$\frac{\Delta P}{\ell}\ [\frac{M}{L^2T^2}]$	$\rho\ [\frac{M}{L^3}]$	$\mu\ [\frac{M}{LT}]$	
$U\ [\frac{L}{T}]$	$D\ [L]$	$\frac{\Delta P}{\ell\rho}\ [\frac{L}{T^2}]$	$\frac{\rho}{\mu}\ [\frac{T}{L^2}]$		Remove M
$\frac{\rho U}{\mu}\ [\frac{1}{L}]$	$D\ [L]$	$\frac{\Delta P}{\ell\rho U^2}\ [\frac{1}{L}]$			Remove T
$\frac{\rho U D}{\mu}$		$\frac{\Delta P D}{\ell\rho U^2}$			Remove L

The final result is often written in the following way,

$$\Delta P = \frac{1}{2}\rho U^2 f(Re)\frac{\ell}{D}$$

where f is called the friction factor and is a function of Reynolds number only $f(Re)$. As with drag coefficients, the friction factor must come from experiments or theory. Note that the Reynolds number is defined using the tube diameter as the appropriate length scale and uses the average fluid velocity, U. Experimental friction factor data for a range of Reynolds numbers is shown in Figure 3.3. Earlier I presented my own data, but here I am taking the data from the professionals that

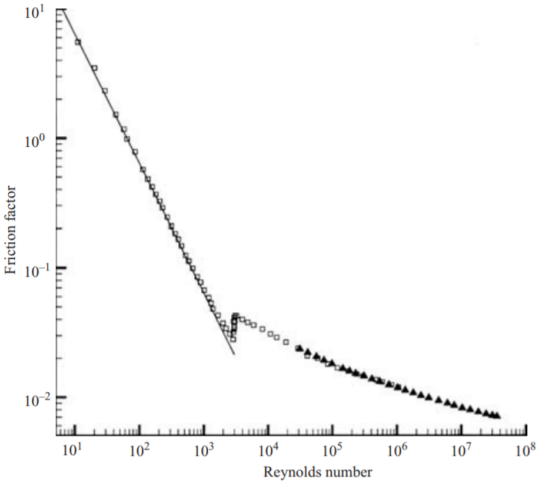


Figure 3.3 Experimental data for friction factor versus Reynolds numbers. Plot is from from McKeon et al. *Journal of Fluid Mechanics*, 2004. The transition from laminar to turbulent flow at Re $\approx$ 2000 is very clearly seen as the jump.

is very high quality. Note that the Reynolds number spans 6 orders of magnitude in Reynolds number.

When the Reynolds number is less than 2300 (this is the most commonly reported transition number), the flow is smooth and laminar and the friction factor can be found analytically,

$$f = \frac{64}{Re} \quad \text{for Re} < 2300.$$

We will compute this solution later in the course. When the Reynolds number is larger than 2300 the flow becomes turbulent and f is found by fitting experimental measurements. There is no analytical solution for the turbulent regime - however the overall friction factor is easy to characterize. It is important to note that the transition around Re $\approx$ 2300 is only seen this cleanly in a careful experiment. In a practical application or more careless experiment (like my own data that I presented in the last chapter) the transition would be seen at a lower Reynolds number and may not be as neatly repeatable as in Figure 3.3.

For turbulent flows, f is also experimentally determined to depend

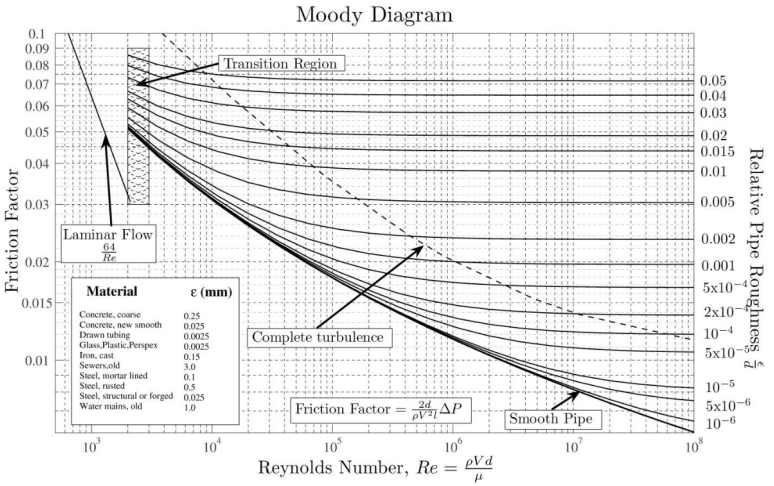


Figure 3.4 Moody diagram for the friction factor in pipe flow. This figure is a reproduction of the way the data are classically presented.

upon roughness of the pipe. In many practical situations the pipe is not perfectly smooth and the roughness of the pipe is critical. The dimensionless term is the relative roughness, ϵ/D, the size of the bumps on the pipe wall relative to the diameter of the pipe. So the general formula from dimensional analysis is

$$\Delta P = \frac{1}{2}\rho U^2 f\left(Re, \frac{\epsilon}{D}\right)\frac{\ell}{D}$$

The diagram in it's classical presentation is reproduced in Figure 3.4 and is called the Moody diagram.

The form of the result is essentially identical to the case of drag on an object. The Reynolds number is the key parameter and the rest of the complexity of fluid flow is wrapped up in the single parameter, f. When the viscosity dominates at low Reynolds number, just as in the drag problem, we have the friction factor varying as $f = 64/Re$. Note that the form of this result can be obtained from dimensional analysis (just as in the drag problem) though the factor of 64 can only come from analysis.

3.3.1 Calculations with the Moody diagram

In order to solve practical problems, we can simply use the Moody diagram as a tool along with the expression,

$$\Delta P = \frac{1}{2}\rho U^2 f(Re)\frac{\ell}{D}.$$

If the velocity of the flow (or volumetric flow rate) is known, then the solution procedure for the needed pressure to drive the flow is straightforward. The procedure is,

- For the given pipe size and fluid, compute the Reynolds number.
- Decide on a good estimate of the pipe roughness.
- Look up $f(Re)$ on the diagram.
- Compute the pressure drop.

If the applied pressure is known, then the solution procedure for the flow velocity is not as straightforward. The procedure is,

- Guess a Reynolds number (you can't compute it since you don't know the velocity).
- Decide on a good estimate of the pipe roughness.
- Look up $f(Re)$ on the Moody diagram.
- Compute the flow velocity as for the known pressure case.
- Compute the Reynolds number for this new flow velocity.
- Look up $f(Re)$ on the Moody diagram.
- Compute the flow velocity again.
- Repeat (if needed) until you get a converged solution.

If at anytime in either procedure (or based on your intuition or knowledge of the problem) you suspect that the flow is laminar with $Re < 2300$, then the Moody chart is not needed. For laminar flow we will later derive an analytical solution $f = 64/Re$, but for now you could consider this to just as equally come from experimental data. For laminar flow we have the simplified relation,

$$\Delta P = \frac{1}{2}\rho U^2 \frac{64}{\mathrm{Re}}\frac{\ell}{D}.$$

or

$$\Delta P = \frac{32\mu\ell}{D^2}U.$$

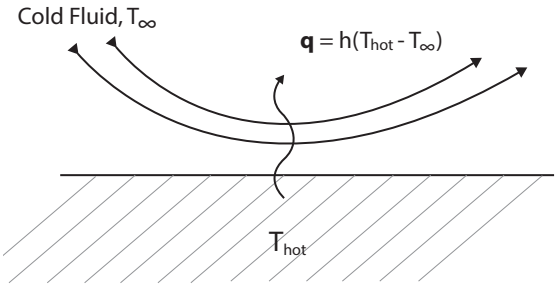


Figure 3.5 Schematic for heat flow from a solid to a fluid by convection. The convection coefficient, h, is usually taken as an empirical parameter.

For laminar flow there is a simple linear relationship between pressure and flow velocity.

3.4 Convection heat transfer

Convection refers to heat transport in a fluid medium when that medium is undergoing net motion, such as when you blow air over your hot soup spoon. Heat flows from the soup to the air directly in contact with the liquid surface and the motion of the air carries the energy away. If a fan is blowing then the rapidly moving and well stirred air will carry more heat away more quickly. If the air is not forced, the hot air will still flow away buoyantly since the hot air is lighter than the surrounding air. The total heat flow in Watts through a surface by convection can be parameterized as

$$Q = hA(T - T_\infty) \tag{3.1}$$

where h is the convection coefficient ($\mathrm{W/m^2K}$), T_∞ is the temperature of the ambient air far from the object, and T is the surface temperature of the object, and A is the surface area. The convection coefficient, h, is usually determined empirically; theoretical determination is not possible except in the simplest geometries and flows. Typical values of h in air are $h = 10 \sim 100$ $\mathrm{W/m^2K}$. While we will deal with convection in more detail in a later chapter, here we will discuss in detail how one could calculate or predict h.

So let's consider a specific problem; a sphere of diameter D is immersed in a fluid flow with velocity U and held at a constant hot temperature T. The fluid is colder at temperature T_∞. How many Watts are put into the sphere in order to maintain the constant hot temperature? As with previous problems on fluid flow, we will assume that the answer depends upon the fluid's density and viscosity. For heat transport are two important thermal properties introduced in the first chapter, the thermal conductivity of the fluid k and the specific heat of the fluid c_p. The thermal conductivity has units of W/mK and the specific heat, c_p, has units of J/kgK. In this case, for subtle reasons we use the constant pressure specific heat.

In dimensional terms we want to find $h = h(D, U, \rho, \mu, k, c_p, T, T_\infty)$. In this case, we now how 4 independent dimensions, mass, length, time and temperature. Our dimensional problem has 9 parameters expressed in 4 independent dimensions, so we have 5 dimensionless parameters. Now it turns out that using some physical intuition (which again, by chapter 3 you are unlikely to have fully formed yet) we can make substantial progress. First since our expression for the total heat flow is

$$Q = hA(T - T_\infty)$$

we can assume that the temperature of the sphere does not impact the convection coefficient. The convection coefficient depends upon the fluid flow around the sphere only. It turns out this can be experimentally tested and found be true for modest temperature differences and flows where the fluid velocity is high (not "modest" and "high" would be defined as dimensionless ratios). If the sphere is quite hot then natural convection can set in where the lower fluid density at the high temperature surface drives flow - which would result in h being a function of T. To apply dimensional analysis, you really have to unlock your inner Lord Rayleigh, use your physical intuition to assume parameters away.

If we assume temperature does not impact the convection coefficient, then we are in a better place as we have $h = h(D, U, \rho, \mu, k, c_p)$ which would give us 3 dimensionless parameters. Working the table yields,

$h\ [\frac{M}{T^3\Theta}]$	$D\ [L]$	$U\ [\frac{L}{T}]$	$\rho\ [\frac{M}{L^3}]$	$\mu\ [\frac{M}{LT}]$	$k\ [\frac{ML}{T^3\Theta}]$	$c_p\ [\frac{L^2}{T^2\Theta}]$
$\frac{h}{k}\ [\frac{1}{L}]$	$D\ [L]$	$U\ [\frac{L}{T}]$	$\rho\ [\frac{M}{L^3}]$	$\mu\ [\frac{M}{LT}]$		$\frac{c_p}{k}\ [\frac{LT}{M}]$
$\frac{h}{k}\ [\frac{1}{L}]$	$D\ [L]$	$U\ [\frac{L}{T}]$		$\frac{\mu}{\rho}\ [\frac{L^2}{T}]$		$\frac{\rho c_p}{k}\ [\frac{T}{L^2}]$
$\frac{h}{k}\ [\frac{1}{L}]$	$D\ [L]$	$\frac{\rho U}{\mu}\ [\frac{1}{L}]$				$\frac{\mu c_p}{k}$
$\frac{hD}{k}$		$\frac{\rho U D}{\mu}\ [\frac{1}{L}]$				$\frac{\mu c_p}{k}$

Of course, the choices I made at each row in the analysis above were arbitrary and there are other equally valid (from a dimensional analysis perspective) choices we could have followed. However, I made choices that led me to the form that is most commonly used in the literature which is

$$\frac{hD}{k} = f\left(\frac{\rho U D}{\mu}, \frac{k}{\mu c_p}\right).$$

Using the standard definitions; $\mathrm{Nu} = hD/k$ is called the Nusselt number and $\mathrm{Pr} = k/\mu c_p$ is called the Prandtl number. Note that the Prandtl number is a material property that only depends on the fluid chosen and not on the geometry or flow conditions. Notice the other parameter is our friend the Reynolds number. Using these final results we have,

$$\mathrm{Nu} = f(\mathrm{Re}, \mathrm{Pr}).$$

For convective heat transfer we have a slightly more complicated result than we had for drag. For drag coefficients, we had them only as a function of Reynolds number whereas here we have a function of two parameters. If we are always using the same fluid (for example, air), then we only need to vary the Reynolds number to fully characterize the heat transfer coefficient since the Prandtl number is a material property.

The final result, though is the same idea as in drag. We can make measurements on a given geometry, but to fully characterize the system we have limited the number of experiments that need to be done when we work in the dimensionless world. We now know that if we wish to compute the heat transfer coefficient for flow over a particular shaped object, we need to solve the problem (numerically or experimentally) as a function of two parameters. We can then make a graph of Nusselt number vs. Reynolds number for different values of the Prandtl number. An example of experimentally determined functional relationship

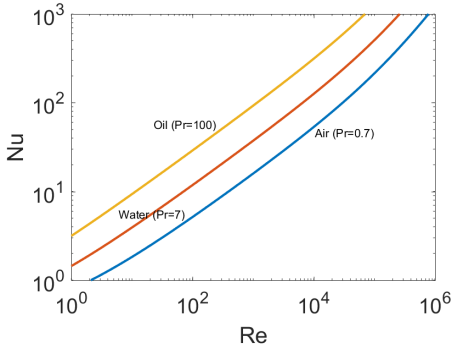


Figure 3.6 Nusselt number as a function of Reynolds number at 3 different Prandtl numbers for flow over a cylinder. The plot is generated from an empirical equation that was fit through experimental data. Note that the data are presented on a log scale and thus nearly follow a power law representation.

is shown in Figure 3.6 This graph (or empirical equation) requires a set of careful experiments to generate, but once the experiments are conducted, they are done for all time. It is much more reasonable to deal with a system which only depends on two parameters than the dimensional formulation. Fortunately, in textbooks on heat transfer, online or in a number of handbooks, you can find correlations or graphs of $\mathrm{Nu} = f(\mathrm{Re}, \mathrm{Pr})$ for a number of common geometries. Even if you have a problem where your geometry is different than anything that was done before, you can usually find something close to give you a reasonable approximation.

As with drag, the choice of the diameter in our case above as the length scale used in the definition of the Reynolds number and the Nusselt number is arbitrary. If I gave you data to use, you need to know what choice I made. I could have chosen the diameter, radius, or circumference. If my geometry is a rectangle, which side of the rectangle should I use? The point here is the same as with drag, if you have data for Nusselt number then the length scale chosen needs to be clearly defined

In practice we will often find that we can collapse the data with some function of the form,

$$\mathrm{Nu} = C\mathrm{Re}^m\mathrm{Pr}^n,$$

where the constants C,m, and n are found from experiments. There often not really any deep meaning into this functional form, it just works well and is simple to use.

It should also be noted that the error bars on heat transfer correlations can be reasonably large. If you use a correlation in a handbook or take data from a graph, you should look at what the stated error is. Even if the correlation is perfect it is common that a real engineering situation may not be identical to the carefully controlled lab setting. As a rule of thumb, if I always keep in my head that I should assume an error of at least 25 %. You will also find that data are often plotted on log scales which hides how far off a given point is because the data might span a few orders of magnitude. Thus, you should never expect heat transfer calculations to be very precise. They can be quite telling, give a good answer for engineering purposes, and simple calculations can inform design. That the calculations are not that accurate is both a blessing and a curse. It is a curse because it means that accurate answers can be hard to obtain. It is a blessing because the proverbial "spherical cow" will often give you an answer that is about as accurate as a more detailed calculation.

3.4.1 Convection calculations

The basic idea for computing heat transfer coefficients becomes quite simple once a correlation exists. In textbooks on heat transfer, one can find correlations which are usually based on experimental data for a number of geometries in external flow. One can easily find relations and data for cylinders, spheres, squares, plates, banks of tubes and so on. Each of these correlations will typically have a range of Reynolds number and Prandtl number where they are valid. You should respect these limits. You should also note the stated accuracy of the correlation if it exists. You also need to be careful in deciding if you need an average or local value of the heat transfer coefficient.

I want to stress that the intellectual magic is contained in the fact that the data will always collapse to some function according to $\mathrm{Nu} = f(\mathrm{Re}, \mathrm{Pr})$. The exact form of the function is not necessarily anything deep. Usually the choice of empirical fit function tries to balance accuracy with simplicity and ease of use.

We have not bothered with the fact that fluid properties can be quite

sensitive to temperature. The simple way to take the variation of fluid properties (viscosity, density, conductivity) into account is to use the values at the average temperature of the object and the free stream fluid. This approach would be the standard method, however, you will sometimes see correlations that have variable fluid properties built into them. The fluid properties variations with temperature is one reason that simple correlations can give answers with large errors.

We will not review the myriad of convective heat transfer correlations that exist. For doing practical calculations, I recommend the book *Fundamentals of Heat and Mass Transfer* by Incroprera and DeWitt. That text (as well as many others) will have a number of correlations for different geometries in forced convection. It is important that you understand any assumptions behind the correlation.

The typical methodology for convection problems is straightforward.

- Identify the geometry and find an appropriate correlation in a text or handbook. Depending on how good of an answer you require, you can often make the "spherical cow" approximation for quick estimates.
- Read the details of the correlation; How accurate is it? Is it good for a limited range of Re or Pr? Make sure you understand if it is appropriate for the problem at hand.
- Make sure you understand what length scale is used in the correlation for the calculation of the parameters. The length scale can be arbitrary so you need to know. For example, for a sphere one could base things off the radius or diameter.
- Evaluate all the fluid properties at the appropriate average temperature, $(T_s + T_\infty)/2$.
- Calculate Re and Pr. Double check that you are in a valid regime for the correlation.
- Use the correlation to calculate Nu.
- Use the fluid properties and geometry to calculate h.
- Do it again from scratch to make sure you believe your number. It is easy to make mistakes. If you have a complicated problem you may want to make a dramatically simplified model first just so you know what order of magnitude to expect.

3.5 Summary

These three examples show how we can combine dimensional analysis with experimental measurements such that we can make accurate predictions. There is no satisfactory "why" embedded in any of these results. However, the results can be used to design and build real devices. Predictions can be made that will be really accurate. I again want to emphasize that we covered only a few examples here and other books provided many more examples.

4
Vector calculus notation and review

Starting in the next chapter we will begin to formulate the laws which govern fluid flow and heat transfer. So far we have relied on observations, experiments, and dimensional analysis to guide us. As we develop our more complete theory, we will find laws will allow us to compute quantities such as the temperature field which can vary both in time and space. Our laws will be "local" in that they can tell us the temperature at a point (in the continuum sense) in space. Inherent in our derivations of these local laws will be volumes, surfaces, and integrals. Using the language of vector calculus will become essential to maintain our results in a form compact enough to provide some physical insight.

I am assuming that upon reading this that you have taken a course in vector calculus but have likely forgotten many of the finer details. This chapter is meant to review the notation and some of the useful theorems. If you have never come across these topics then my coverage here will not be sufficient. Our use of vector calculus will be as the language we use to describe the physics. We will not need to carry out complicated surface and volume integrals, however, we will need to come to understand an intuitive and physical level what a surface or volume integral *is*.

4.1 Scalar and vector functions

Many physical things we will want to describe are scalar functions. A scalar function means that there is a single value at every location in space. That value at each point can also change with time, but the field

has no sense of direction. Two examples of scalar functions that we will discuss quite a bit are temperature, T, and pressure, P. When we are being explicit, we will write scalar functions as $T(x, y, z, t)$ to denote that temperature can vary at every location in 3D space and evolve with time. In two dimensional problems, we will most commonly visualize the scalar function with a contour plot, where lines show values of constant temperature. The contour plot is just like reading a topographic map; follow a contour line and your elevation doesn't change. Follow an contour line of a temperature field and the temperature does not change along the path.

We will also want to describe quantities that have directionality to them and these will be vector functions. We will use a boldface notation for vector functions. The most common quantity we will discuss is a fluid flow's velocity field, $\mathbf{v}$. If we are being explicit we will sometimes write $\mathbf{v}(x, y, z, t)$. In two dimensional problems we will visualize the velocity field with a set of arrows at many locations in space, showing the velocity vector's magnitude and direction. The velocity vector has three components, thus it is important to always remember that the vector function is comprised of three scalar functions for each component. For velocity, we will commonly use the notation u, v and w for the x, y, and z components respectively. We will sometimes write out vector functions as a column vector

$$\mathbf{v} = \begin{bmatrix} u \\ v \\ w \end{bmatrix},$$

or sometimes as components using unit vectors,

$$\mathbf{v} = u\mathbf{i} + v\mathbf{j} + w\mathbf{k}$$

Another example of a common vector function that we will describe is the heat flux, $\mathbf{q}$. Heat flux will describe the flow of heat (in terms of Watts per unit area) - which has both magnitude and direction. Heat flux will be discussed in detail in the next chapter.

Examples of scalar and vector fields in a 2D world are shown in Figure 4.1 in the context of weather. The contour map of atmospheric pressure is shown at one instant in time during a classic "Nor'easter" snowstorm that is common in New England. The low pressure contour off the coast generates a strong counterclockwise wind that brings moisture in from

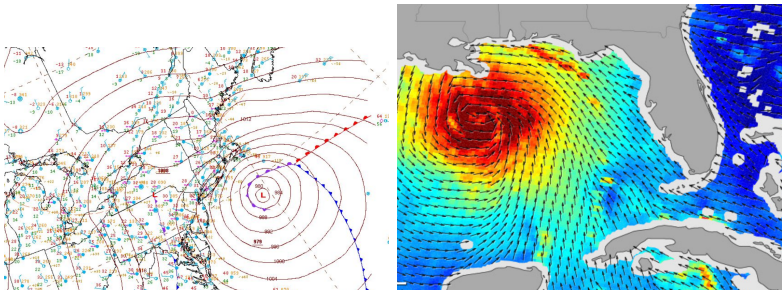


Figure 4.1 Examples of scalar and vector functions in weather applications. On the left are pressure contours during a classic Nor'easter snowstorm (image from the National Weather Service archives). On the right are velocity vectors from Hurricane Katrina in 2005 (from Nasa's Earth Observatory). The velocity vectors are placed on top of an image of another scalar field, which is the magnitude of the velocity at that point.

the ocean. The image of velocity vectors from Hurricane Katrina in 2005. The color image shows the magnitude of the velocity field (a scalar field) and the vectors are superimposed on top.

4.2 Gradient, divergence, and curl

There are three operations that we will use repeatedly in our study are divergence, gradient and curl. These three operations that derivative with respect to space. While all three have some similarities we will think of them all as pretty distinct physical things. For all three, it is convenient to use the notation for "del", with the symbol ∇, written as a column vector,

$$\nabla = \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix},$$

or equivalently using unit vectors,

$$\nabla = \frac{\partial}{\partial x}\mathbf{i} + \frac{\partial}{\partial y}\mathbf{j} + \frac{\partial}{\partial z}\mathbf{k},$$

where $\mathbf{i}, \mathbf{j}, \mathbf{k}$ are the unit vectors in the x, y, z direction.

When we apply "del" to a scalar function (T in the example below) we define the *gradient* which gives back a vector,

$$\nabla T = \begin{bmatrix} \frac{\partial T}{\partial x} \\ \frac{\partial T}{\partial y} \\ \frac{\partial T}{\partial z} \end{bmatrix},$$

or equivalently using unit vectors,

$$\nabla T = \frac{\partial T}{\partial x}\mathbf{i} + \frac{\partial T}{\partial y}\mathbf{j} + \frac{\partial T}{\partial z}\mathbf{k}.$$

Remember, *the gradient of a scalar field gives a vector*. We can also take a gradient of a vector field, but let's leave that issue to the side for now. The most important property of the gradient is that the resulting vector field always points perpendicular to the contours and points "uphill". If you see closely spaced contour lines on a topographical map you know the terrain is very steep. If you see closely spaced isothermal lines, then you know the temperature gradient will have a large magnitude at these locations.

When we take the dot product of "del" with a vector field we define the *divergence* which gives back a scalar field. The divergence of the velocity vector is defined as,

$$\nabla \cdot \mathbf{v} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}.$$

The divergence is a measure of how much a vector field is, well, diverging. If you see a point in the vector field where all the velocity vectors were flowing out of a point, this would have positive divergence. If all the vectors were flowing into a point, then that would have negative divergence. If some vectors were flowing in and some out, and if they were in perfect balance there would be zero divergence. The divergence is shown schematically in Figure 4.2. We will see that divergence is connected to the idea of conservation. In a fluid flow if there is non zero divergence of the velocity field, then it means that mass is accumulating at that point. Non-zero divergence can only occur in a compressible flow. Remember, *the divergence of a vector field gives a scalar field. It does not make sense to take the divergence of a scalar field.*

Our final operator of interest occurs when we take the cross product of "del" with a vector. This operation is defined as the curl. For

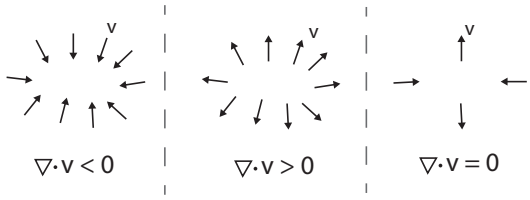


Figure 4.2 Schematic of the divergence of velocity.

example, the curl of the velocity field is (known by the symbol ω),

$$\omega = \nabla \times \mathbf{v} = \begin{bmatrix} \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \\ \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \\ \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \end{bmatrix}.$$

The curl has to do with whether the vector field is spinning or not. We need to be a little careful, we can have circular looking vector fields that have zero curl. The physical interpretation of curl will be put to the side for now and we will dig into a little deeper later in the book. *The curl of a vector results in a vector. It does not make sense to take the curl of a scalar.*

It is important to remember that del has units of 1/Length; for example the units of ∇T are Kelvin/meter. If we work in dimensionless terms we would very often (but not always!) decide to scale the x, y, and z coordinate directions by the same length scale L. In this case our dimensionless del would become,

$$\nabla = \frac{1}{L}\tilde{\nabla}.$$

Note that the same units apply to divergence, gradient and curl.

4.3 Normal vectors and flow through a surface

Consider an arbitrary surface. This could be a surface of a distinct piece of material, i.e. the outer skin of a ball. We could also be taking an imaginary slice or cross section through a solid body. Whenever we define a surface, we also need to define a normal vector. A normal

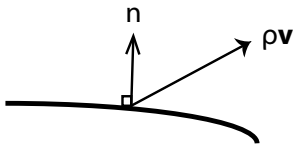


Figure 4.3 Schematic of the normal vector and mass flux at a surface. Here the vector field and the normal vector are not aligned so only some fraction of the total mass flux at this location is cross the surface.

vector is a unit vector (magnitude of 1) which points perpendicular to a surface. When we have a closed surface, by convention we point the normal vector outward. If we don't have a closed surface, then we need to define which direction the normal vector points.

We will commonly take the dot product of a vector field with the normal vector of a surface. This product provides the the local rate that "stuff" is flowing across the surface. For example, consider the vector field $\rho\mathbf{v}$ where ρ is the fluid density and $\mathbf{v}$ is the velocity field. The units are $\mathrm{kg/s \cdot m^2}$. The meaning is the local mass flux, the rate that fluid mass is being carried by the velocity field per unit area. The dot product $\rho\mathbf{v} \cdot \mathbf{n}$ gives the rate, per unit area, that mass is crossing the surface at that point. If the mass flux vector field is aligned with the normal vector of the surface, then all the matter that is moving is also crossing the surface. If the flow is perpendicular to the surface then $\rho\mathbf{v} \cdot \mathbf{n} = 0$ and nothing is crossing the surface even things are flowing. The dot product gives us a measure of how well aligned the flow is to crossing the surface. The surface normal and mass flux are shown schematically in Figure 4.3.

4.4 Volume integrals

Imagine we have an arbitrary volume of material and we break the volume up into a bunch of little discrete "bricks". We weigh each brick, sum them up and we have the total mass of the volume; $M = \sum m_i$. We can also compute the mass of an individual brick by multiplying the density by the volume, $m_i = \rho_i V_i$. Now, you should be familiar with

the idea that if we break the such a sum up into smaller and smaller bricks that in the limit of lots of little pieces we have an integral,

$$M = \int \rho dV.$$

What we mean by this notation is that we integrate over x, y, and z;

$$M = \iiint \rho(x, y, z) dx dy dz.$$

We will typically use the former notation for simplicity. If we actually want to compute the integral above, we would need the density as a function of x, y, and z as a known. If the density is a constant then we can pull the density out of the integral in which case $V = \int dV$; just the total volume of the region.

We will need to remember that the element dV is a little volume element, like a single brick in the wall. We also need to always keep in mind that dV has units of volume. Above when we integrate mass density (mass per unit volume) over the volume we are left with total mass; the units of density turned to mass. Most often we will use the volume integral to count up how much total "stuff" is contained in a volume of material. By "stuff" we will mean total mass, energy, or momentum. These type of total volume integrals are crucial in our conservation laws.

We will also use the volume integral to give us a measure of the average value of some function, for example one useful average temperature of an object could be computed as,

$$T_{\text{ave}} = \frac{1}{V} \int T dV.$$

Volume integrals can be computed of either scalar or vector fields. The volume integral of a scalar field is a single number and the the volume integral of a 3D vector field is a single 3D vector.

4.5 Surface integrals

In addition to volume integrals, we will also very commonly describe physical things using surface integrals. The concept is in many ways similar to the volume integral only now we think about little patches

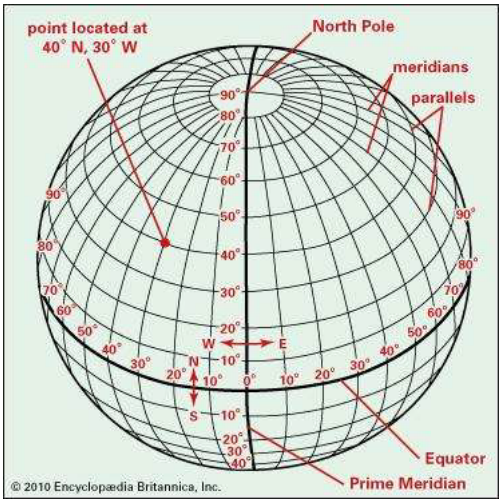


Figure 4.4 Latitude and longitude grid for the earth (from Encyclopedia Britannica).

of surface area. Take for example the grid of latitude and longitude on the globe shown in Figure 4.4. The grid patches are not equal in area, thus we would need to weight any sort of sum by the area of each patch. This weighting of the different area patches in a sum is what the expression for the surface integral naturally does for us. We will use surface integrals in many ways.

One example is the average of some scalar function on the surface. If we wanted to compute the average surface temperature of the earth, for example, we would express the calculation as,

$$T_{\text{ave}} = \frac{1}{A} \int T dS,$$

where T varies at every point along the surface.

Very often we will compute the surface integral of a quantity which has some dot product with the surface normal. These expressions will be useful for telling us how much "stuff" is crossing a surface. For example $\rho \mathbf{v}$ gives the total mass flux (in units of $\text{kg/s} \cdot \text{m}^2$), $\rho \mathbf{v} \cdot \mathbf{n}$ gives

the how aligned the mass flux is with normal vector, and

$$\int \rho \mathbf{v} \cdot \mathbf{n} dS,$$

would give the total mass flow (in units of kg/s) through the surface.

We will very often apply the surface integral to an entire closed surface which encases some volume of material. A more formal notation for the surface integral of a close surface would be,

$$\oiint \rho \mathbf{v} \cdot \mathbf{n} dS.$$

This notation is explicit because it uses two integrals to highlight that we are integrating over an area. The circle in the surface integral is to remind us that we are integrating over a closed area which bounds the volume. To keep our notation simpler (as with the volume integral) we will not use the full notation with two integrals.

Can also compute surface integrals for vector quantities. Later in the course we will discuss pressure a lot. In a fluid, pressure is an inward internal force of the fluid. Pressure has units of force per unit area. Pressure multiplied by the surface normal gives a stress vector, a force per unit area acted at a point along the surface. If we add up all the little stress vectors and weight them by the area over which they locally act, we have the total force due to pressure acting on this surface. Thus, the net force due to pressure.

$$\mathbf{F} = \int P \mathbf{n} dS$$

The volume integral of a scalar field is a single number and the the volume integral of a 3D vector field is a single 3D vector.

4.6 Surfaces and volumes in 2D

Through this course I will often use 2D problems as examples since they are illustrative, easy to sketch, and easier to solve than 3D. I am a poor artist and a little lazy, so 2D works great for me. It is often a point of confusion as what is a 2D volume? What is a 2D surface? Simply put, a 2D volume is a plane that we can sketch on a piece of paper and a 2D surface is the line that encloses that plane. When we think

about what this means for our 3D world, just imagine that the thing you draw extends outward and into the page forever. The 2D object that we draw is a cross section of an infinitely long object. Working in 2D can sometimes confuse our units, so you have to remember that we are computing things per units depth into the page.

4.7 Divergence theorem

The primary vector calculus theorem we will use is the divergence theorem, which states for any vector field $\mathbf{v}$,

$$\int \nabla \cdot \mathbf{v} dV = \int \mathbf{v} \cdot \mathbf{n} dS.$$

Even though we are using $\mathbf{v}$ as our vector field here, the above result is a general result from calculus and has nothing to do with whether $\mathbf{v}$ is a physical velocity.

We can derive this result by sticking to a simple 2D example using vector field $\mathbf{v}$ with x and y components u and v. The components of the vector field are functions of space; $u(x, y)$ and $v(x, y)$. For simplicity we will take our region of interest to be a unit square. We will start by evaluating the surface integral. On the left boundary $\mathbf{v} \cdot \mathbf{n} = -u$ and on the right boundary $\mathbf{v} \cdot \mathbf{n} = u$. The change in sign is that on the left the normal vector and the positive x component of the velocity vector point in opposite directions whereas on the right they point in the same direction. The dot product means that the sign will be opposite for the two terms. The same holds for the upper and lower surfaces, only here it is the y component of the velocity that shows up from $\mathbf{v} \cdot \mathbf{n}$. At this point,

$$\int \mathbf{v} \cdot \mathbf{n} dS = -\int_{y=0}^{y=1} u(x = 0, y) dy + \int_{y=0}^{y=1} u(x = 1, y) dy$$

$$-\int_{x=0}^{x=1} v(x, y = 0) dy + \int_{x=0}^{x=1} v(x, y = 1) dx$$

or by grouping terms,

$$\int \mathbf{v} \cdot \mathbf{n} dS = \int_{y=0}^{y=1} \left(u(x = 1, y) - u(x = 0, y) \right) dy$$

$$+\int_{x=0}^{x=1}\left(v(x,y=1)-v(x,y=0)\right)dx.$$

Recall that the fundamental theorem of calculus states that for any function,

$$u(x=1)-u(x=0)=\int_{x=0}^{x=1}\frac{\partial u}{\partial x}dx.$$

Therefore, our first expression can be rewritten as

$$\int \mathbf{v}\cdot\mathbf{n}dS=\int_{y=0}^{y=1}\left(\int_{x=0}^{x=1}\frac{\partial u}{\partial x}dx\right)dy+\int_{x=0}^{x=1}\left(\int_{y=0}^{y=1}\frac{\partial v}{\partial y}dy\right)dx.$$

Since the order of integration doesn't matter,

$$\int \mathbf{v}\cdot\mathbf{n}dS=\int_{y=0}^{y=1}\int_{x=0}^{x=1}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)dxdy.$$

Therefore, we can write the final result in our vector calculus language,

$$\int \mathbf{v}\cdot\mathbf{n}dS=\int \nabla\cdot\mathbf{v}dV.$$

Note that we did this derivation for a unit square in 2D. We could have been a little more general and the result would have held up. This is one of the most useful theorems in this course. The theorem relates the total flux of a vector field through a closed surface to the divergence of the same field inside the volume.

4.8 Summary

As stated in the beginning, I am assuming that the notation is familiar to you and you are not completely new to vector calculus. If you are new to the topic, then this "introduction" is going to be too brief. You probably noticed that I focused more on the physical interpretation of various surface and volume integrals rather than actually *computing* surface and volume integrals. This focus on the *meaning* will continue through the book. We will use the language of vector calculus to describe our fundamental conservation laws and the notation is used throughout the derivations.

5
Conduction heat transfer

We will now begin deriving some of our fundamental laws by starting with the example of heat conduction in a solid. Despite the fact that in the introduction we discussed all the exciting applications which involve fluid flow, we start with the example of the transport of heat with no fluid flow. The reasons for this are two fold. First, the problem of heat conduction is important in a number of applications that we will discuss as we proceed. Second, problems with no fluid flow are easier. Starting with heat transport in stationary media allows us to work through the basic mathematics used to describe flow and transport processes. Later, we will add the complexity of fluid flow on the transport of heat.

Heat conduction will be our first example of *diffusion*; specifically the diffusion of heat. In subsequent chapters we will consider the diffusion of mass and momentum. Mathematically and physically, all diffusive process have some similarities, so this chapter will serve as a prerequisite for later chapters that discuss diffusive processes.

5.1 Heat flow by conduction

In analyzing thermal systems, we will constantly discuss heat flow, q (Watts), through our systems. In order to describe the flow of heat, the first quantity we define is the heat flux vector, $\mathbf{q}$. The boldface notation denotes that this is a vector quantity. Heat flux has magnitude and the direction points from hot to cold and points in the direction that heat flows. The heat flux vector has units of W/m^2. It is the rate that energy is being transported per unit area. If I measured the heat flux on a wall

of a warm building on a cold day and found that the value was the same at every location on the wall. I would multiply the heat flux by the area of the wall to compute how many Watts I was losing through the wall.

I should comment on the use of the term "heat flow". When there is "heat flow" between a hot and a cold object, no actual substance is flowing between the objects. Long ago, people thought there was something flowing, an invisible weightless fluid called the caloric, which was thought to pass between objects in a conserved way. James Joule (1818-1889) is typically credited with conducting many careful experiments to show that heat and mechanical work are both forms of energy. The work of Joule and others effectively showed that the caloric theory of heat was wrong. Joule's experiments helped demonstrate that there is a quantity we call energy which has the property that it is conserved. Conservation of energy is known as the First Law of Thermodynamics. If the energy of a cooling object decreases, the energy of the surroundings must increase to conserve energy. Thus, heat flow is really energy transfer. Even though it has been a century since the caloric fluid theory has been dismissed, we still use the term heat flow since it still has some intuitive appeal. In fact, the equations of this chapter could be derived based on a caloric fluid theory since we are ignoring mechanical work for now.

Thermal energy can be transferred between objects by three distinct physical mechanisms; conduction, convection, and radiation. The physics of these three processes are distinct. The focus of this chapter will be **Conduction**, which is the transport of heat through a medium where there is no motion of the medium itself (as in a solid). An example would be sticking a metal spoon in a hot pot of water; if you hold the handle too long the metal heats up and burns your hand. Heat is conducted through the metal in the spoon.

Conduction is inherently a molecular process. The details depend upon whether the medium is a solid, liquid, or gas. Perhaps it is easiest to visualize heat conduction in a stationary gas, shown schematically in Figure 5.1. In a gas, temperature corresponds to molecular motion. While on average the gas may be stationary, individual molecules are whizzing around in all directions. In air at room temperature, the average translational speed of a single molecule is on the order of 400 m/s. It is the average kinetic energy of the molecules at the molecular level

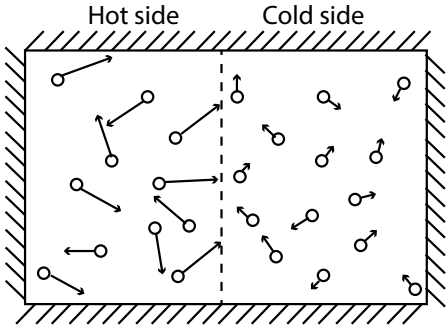


Figure 5.1 Schematic of heat conduction in a gas. When a fast moving hot molecule collides with a slow moving cold molecule then the cold molecule speeds up and the hot one slows done. Eventually all the molecules inside the box undergo enough collisions that the temperature is uniform throughout the box.

that corresponds to the macroscopic quantity that we call temperature. Imagine we have hot and cold gas in a closed, insulated container where the two regions are initially separated by a partition. When we remove the partition, the faster molecules in the hot region fly over and collide with slower moving molecules in the cold region. If you think of the molecules as billiard balls then upon collision, the hot molecule will slow down (cool down) and the cold molecule will speed up (heat up). As time goes by, molecular collisions between fast and slow molecules will equilibrate the system. After a long time, all the molecules have the same average translational speeds and thus the temperature is uniform throughout. In a solid, the process is conceptually the same, only the molecular picture is perhaps more unfamiliar.

When we have a system at steady state, heat is transported by conduction across a material by the following equation,

$$q = \frac{k}{L}\Delta T \tag{5.1}$$

where ΔT is the temperature difference, k is thermal conductivity (W/m K) which is a material property, and L is the distance over which the temperature difference occurs. This law is called Fourier's law of conduction, named after Jean-Baptiste Joseph Fourier (1768-1830) a French physicist, mathematician, and scientific advisor to Napoleon. In

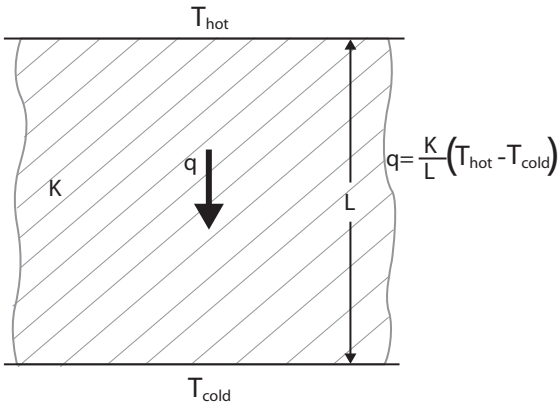


Figure 5.2 Schematic of 1D heat conduction and the use of Fourier's law at steady state. The heat flux is proportional to the temperature difference across the material.

many cases we can assume that k is constant, though this is not true for all materials. In some cases thermal conductivity can be a strong function of temperature. For highly conductive materials such as copper, $k = 300$ W/m K while materials such as glass might have $k = 1$ W/m K. We will return to a generalization to this law soon. It is important to note that Fourier's Law is not really a "law". It is the experimentally observed behavior of many materials, but it is not a fundamental law like conservation of energy.

5.1.1 Other modes of heat transfer

For the purposes of studying conduction in a solid, the other two modes of heat transfer only occur at the surface of a solid. Convection and radiation are common *boundary conditions* in heat conduction problems. **Convection** refers to heat transport in a fluid medium when that medium is undergoing net motion. This mode of heat transfer was discussed in Chapter 3. Heat flux through a surface by convection is be described by

$$q = h(T - T_\infty) \tag{5.2}$$

where h is the convection coefficient ($\mathrm{W/m^2K}$), T_∞ is the temperature of the ambient air far from the object, and T is the surface temperature of the object. For the purposes of heat conduction in a solid, the coefficient, h, will be treated as a known quantity - though as alluded to previously it is not always an easy number to know!

Heat transfer by **radiation** is energy transported by electromagnetic waves. A key difference with radiation heat transfer from convection and conduction is that no medium is needed. The most familiar example of radiation is the energy that the earth receives from the sun. All objects above absolute zero emit thermal radiation and the wavelength of the emitted radiation depends upon the temperature. What we call light is the same thing as thermal radiation. Our eyes are tuned to detect thermal radiation in a certain range of wavelengths, which corresponds to the mean wavelength of radiation emitted from the sun. The net heat flux between an object exchanging radiation with its surroundings is,

$$q = \sigma(T^4 - T_{sur}^4) \tag{5.3}$$

where $\sigma = 5.67 \times 10^{-8}$ $\mathrm{W/m^2K^4}$ is the Stefan-Boltzmann constant and T_{sur} is the temperature of the surroundings. Radiation heat transfer is most important at very high temperatures as seen by the increase in the heat flow with the fourth power of the temperature. There are many interesting aspects of radiation heat transfer that we will not consider in this course. It is useful to be familiar with the radiation equation, we will not discuss it further in this book.

5.2 One-dimensional heat equation and conservation of energy

We will now derive the governing equation for heat conduction in a one-dimensional solid. We will soon generalize this derivation but will try to start simple so that we can become familiar with some of the concepts.

You have probably heard that energy is conserved. This seemingly simple concept is actually quite deep and is something we will discuss in more detail in the chapter on Thermodynamics. For now, we will keep things simple. To quantify this basic conservation law, we consider an

arbitrary material volume and write the law as,

$$\Delta E_{\text{stored}} = E_{\text{in}} - E_{\text{out}}. \tag{5.4}$$

The change in energy stored ΔE_{stored} in an arbitrary volume equals the energy that crossed the surface coming into the volume minus the energy crossing the surface going out of the volume. Conservation of energy is not something we derive, but something we state to be true because every experiment we have ever thought to do confirms this interpretation.

In this chapter, we are working with restricted version of this law and are only considering thermal energy. We will not consider that any mechanical work is done and we will not consider thermal energy generated inside the volume which may occur by chemical or nuclear reactions. In many cases we will find it more useful to use conservation of energy in rate form which we would write as,

$$\frac{dE_{\text{stored}}}{dt} = \dot{E}_{\text{in}} - \dot{E}_{\text{out}} = \dot{E}_{\text{net}}. \tag{5.5}$$

The rate equation says we simply need to look at the net rate that energy is coming in and out of our volume and that must be equivalent to the time rate of change of the energy stored inside the volume. I should comment on the use of $\dot{E}_{\text{in}}$ to denote the *rate* of energy coming into our volume. It is common notation to use a dot above a variable to denote a rate. This dot should not be confused with the one often used in mathematics books where $df/dt \equiv f'(t) \equiv \dot{f}(t)$.

Energy has units of Joules and rate that energy crosses the surface, $\dot{E}$, has units of Watts. Remember that a Joule is the same as Newton-meter.

5.2.1 Storage of thermal energy

You may recall from earlier physics courses (and the beginning of this book) an equation $\Delta E = mc\Delta T$. This equation, describes the change in thermal energy stored in a solid. ΔE is the change in energy (J), m is the mass (kg), c is the specific heat (J/kg$K$), and ΔT is the change in temperature. The Δ is used to remind us that only changes in energy matter. The change in thermal energy stored in a solid is proportional to the change in temperature. Energy is always talked about in terms of

a change and what we define as the zero reference for energy is arbitrary. The fact that energy has an arbitrary reference should be familiar, it is just like we can always define height=0 anywhere we like in problems with gravitational potential energy. The equation for energy storage is only valid when the temperature of the body is constant throughout.

5.2.2 Heat flow through a wall

Let's consider heat flow through a wall which is very large in two of the dimensions (that we don't care about), with respect to its thickness (the dimension we do care about). Imagine a wall in your home and wanting to understand how heat flows from the warm interior to the cold exterior on a cold winter day. It seems reasonable to assume the heat flow is mostly one-dimensional. We want to understand how heat flows in and out of this system, what the temperature distribution is, and we may want to answer a number of important questions. How long does it take take energy to move through the wall? What parameters does the rate of temperature change depend upon? For fixed conditions, what is the final steady state temperature distribution?

The amount of stored thermal energy inside the wall (relative to some reference temperature) is ,

$$E = mc(T - T_{\text{ref}})$$

where m is the mass, c is the specific heat, T is the temperature of the wall, and T_{ref} is the arbitrary reference temperature. However, the above expression assumes T is constant throughout the wall and we want to consider a case where the temperature is not constant in space. Imagine we could chop up the wall of length L into N thin slices of size $dx = L/N$ in size and assume each slice was at a different temperature. In this case the stored thermal energy would be the sum of all slices,

$$E = \sum_{i=1}^{N} m_i c(T_i - T_{\text{ref}})$$

where the subscript i refers to the i^{th} section. If the size and mass of each section is the same then $m_i = m/N$ and in the limit of $N \to \infty$ the sum becomes the integral,

$$E = \frac{mc}{L} \int_0^L (T(x,t) - T_{\text{ref}}) dx.$$

Normally for an integral for a total quantity we write in terms of the materials mass density, ρ. Since the total mass of the wall is $m = \rho AL$, then

$$E = A \int_0^L \rho c(T(x,t) - T_{\text{ref}}) dx, \tag{5.6}$$

where A is the wall's cross sectional area. You should confirm the units of this expression and convince yourself the units are Joules. The above form is the one we will always use, where our quantity is give on a per unit volume basis and then we integrate over a volume.

Inside the wall, there is an internal heat flux $q(x,t)$ (W/m^2) that flows from hot to cold within the wall. The local heat flux is different at every location and can evolve with time. The balance of energy for the entire wall simply states that the rate of change of the energy stored in the wall is equal to the net rate of heat passing in from the left at $x = 0$ minus what goes out at $x = L$. Since heat flux has direction, positive heat flux points from left to right. The balance of energy in rate form is,

$$\frac{dE}{dt} = A(q(0,t) - q(L,t)), \tag{5.7}$$

where A is the cross sectional area of the wall. Using our expression for the energy stored we obtain,

$$\frac{d}{dt}\left[A \int_0^L \rho c(T(x,t) - T_{\text{ref}}) dx\right] = A(q(0,t) - q(L,t)). \tag{5.8}$$

Now let's assume the case which is common for solids where is the density and specific heat are just constants. Since the reference temperature is a constant with time we can remove it from the equation when we take the time derivative. We also cancel the cross sectional area from both sides,

$$\frac{d}{dt} \int_0^L \rho c T(x,t) dx = q(0,t) - q(L,t). \tag{5.9}$$

We can now use the Fundamental Theorem of Calculus to rewrite the right hand side of the equation as,

$$\frac{d}{dt} \int_0^L \rho c T(x,t) dx = -\int_0^L \frac{\partial q}{\partial x} dx.$$

Note that we used the Fundamental Theorem of Calculus "backward"

from how we are probably used to thinking about it. Since the region of integration is not changing with time and we have constant material properties the order of integration and differentiation on the left side of the equation doesn't matter,

$$\int_0^L \left(\rho c \frac{\partial T}{\partial t} + \frac{\partial q}{\partial x} \right) dx = 0.$$

You might wonder why I changed the d/dt to a $\partial/\partial t$ above. When I used d/dt I meant taking the time derivative of integral expression which is only a function of time. Partial derivative notation is used in the final equation since the local value of the temperature is function of time and space, $T(x,t)$.

Putting the equation in this final integral form allows us to turn the expression into a local differential equation. The equation above says the integral of some quantity in parenthesis must equal to zero. An integral can be equal to zero when the thing you are integrating is non-zero. Think about sin(x) integrated from 0 to 2π. The function is positive some of the time and negative the rest and in perfect balance. However, in our problem the volume of integration is arbitrary. While we derived it for the wall from $0 < x < L$ we would obtain the same equation if we derived the equation from $0 < x < L/2$. I could make the integration region bigger, smaller or move it. Thus the only way the integral can *always* be zero is if the thing we are integrating is zero. Everywhere. Therefore our integral equation can be written in differential form as

$$\rho c \frac{\partial T}{\partial t} + \frac{\partial q}{\partial x} = 0. \tag{5.10}$$

This equation must hold at every point within the wall.

This equation says that if the heat flux is a positive but decreasing function with x at a point, then the local temperature is increasing with time. If the heat flux is locally a positive decreasing function, then more heat is going into a point than leaving it meaning that the local storage of energy (or temperature) must go up. If the heat flux is constant in space then the temperature does not change.

5.2.3 Fourier's Law

At this point our conservation of energy equation is useless. There is only one equation but there are two unknowns, the heat flux and temperature. What is needed to close the problem is a relationship between two. This closure is provided by Fourier's law, which is really an empirical statement and not a law at all. It can be derived rigorously from molecular theories for some materials. Fourier's law is our first example of a constitutive law.

Previously we stated Fourier's Law in a simple form which is reminiscent of Ohm's law in circuits,

$$q = \frac{k}{L}\Delta T. \tag{5.11}$$

If we think of heat flow as current and temperature difference as voltage drop, then the thermal resistance is L/k. We will return to the analogy with resistors circuits in the next chapter.

A more general form of Fourier's law for continuous functions states,

$$q = -k\frac{\partial T}{\partial x}. \tag{5.12}$$

The law states there is a linear relationship between heat flux and temperature gradient. The constant, k, is called the thermal conductivity. It is a material property that we can look up in a book (or wikipedia). Also consider the sign in Fourier's law. If the temperature is going from hot to cold as you move left to right, then the derivative of the temperature with respect to x is a decreasing function. Therefore, the heat flux vector points from left to right.

As with the specific heat, the thermal conductivity is related to the molecular nature of the material. For some simple materials such as gases, there are molecular theories which allow us to compute the thermal conductivity from essentially first principles. It is also a property that can be measured. In this course we will treat it as a material constant which we know and can look up.

5.2.4 Heat equation

Using Fourier's law and our conservation equation we can combine them to obtain,

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x}\right). \tag{5.13}$$

Assuming that the density, thermal conductivity, and specific heat are constants and using the definition of thermal diffusivity as $\alpha = k/\rho c$ we obtain a final equation as,

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}. \tag{5.14}$$

The property α is a material property that has units of m^2/s. This property is tabulated for most materials and has a range of values. For example, copper = 1.17 cm^2/s and building insulation 0.001 cm^2/s.

The equation is a partial differential equation which relates changes in temperature with respect to time and space. The equation says the first derivative of temperature in time is proportional to the second derivative in space. The second derivative in space is the curvature. If the spatial temperature field is locally curved, then the temperature at that point will change with time. This behavior is shown graphically in Figure 5.3. Here we show three temperature profiles in space at a snapshot in time. One profile is linear while the others are concave up and concave down. Lets consider the temperature at the center point in space. In the concave upward case, curvature at the center is positive. The center temperature therefore increases with time. Physically this make sense because the material a little to the left and a little to the right is hotter than the center, pulling the center temperature upwards. The situation is reversed for the concave down example. When the temperature gradient is linear, the point to the right is hotter (pulling up) and the temperature to the left is cooler (pulling down) and thus the temperature remains constant with time.

To solve the heat equation, we need boundary conditions and an initial condition. The heat equation only describes the evolution of the temperature inside a region, we must know how this region is attached to the world around it. Since the equation has a second derivative in the spatial coordinate, we need two boundary conditions - one at each end of the domain. Further, since the heat equation describes the evolution

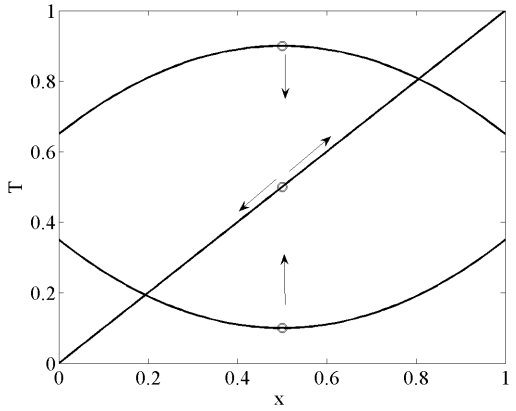


Figure 5.3 Graphical interpretation of the heat equation. We show three sample spatial temperature profiles at a particular instant in time. Consider whether the temperature at the center (x=0.5) increases, decreases, or stays the same (in time) based on the spatial curvature.

of the temperature field we need to know what the initial temperature distribution in the domain. We will discuss different boundary conditions in a later section.

5.3 Dimensional analysis

Now let us consider a specific 1D problem. Consider a 1D object is at a uniform hot temperature T_h, and we suddenly cool the two ends by pulling them down to a colder temperature T_c. The wall has a total thickness of ℓ. For the problem of interest, the equation, initial condition and two boundary conditions would be stated as

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \tag{5.15}$$

$$T(x, t = 0) = T_h;$$

$$T(x = 0, t > 0) = T_c; \qquad T(x = \ell, t > 0) = T_c. \tag{5.16}$$

We want to understand the rate at which this wall cools.

5.3.1 Pi Theorem

If we didn't know the equation but only knew the parameters of the problem we would say the parameters are the temperature T, the spatial coordinate x, the time t, the initial temperature T_h, the temperature at the boundaries, T_c, the size of the object ℓ, the density ρ, the thermal conductivity k, the specific heat C. The units of the problem are mass $[M]$, length $[L]$, time $[T]$, and temperature $[\theta]$. Thus we have 9 parameters and 4 independent dimensions, and 5 dimensionless parameters. Since temperature difference is the only thing that usually matters, we could probably argue that there are two temperatures that matter $T - T_c$ and $T_h - T_c$, though we are still left with 4 dimensionless parameters. If we wrote out the table we would have

$T - T_c$ $[\theta]$	x [L]	t [T]	$T_h - T_c$ $[\theta]$	ℓ $[L]$	ρ $[\frac{M}{L^3}]$	c $[\frac{L^2}{T^2\theta}]$	k $[\frac{ML}{T^3\theta}]$

We can now proceed to remove dimensions from the problem. Since mass appears in only two parameters, we can easily remove mass from the problem,

$T - T_c$ $[\theta]$	x [L]	t [T]	$T_h - T_c$ $[\theta]$	ℓ $[L]$	ρ $[\frac{M}{L^3}]$	c $[\frac{L^2}{T^2\theta}]$	k $[\frac{ML}{T^3\theta}]$
$T - T_c$ $[\theta]$	x [L]	t [T]	$T_h - T_c$ $[\theta]$	ℓ $[L]$		c $[\frac{L^2}{T^2\theta}]$	$\frac{k}{\rho}$ $[\frac{L^4}{T^3\theta}]$

The next step might be less obvious. We have a number of choices to remove the other units. It may not clear where we should go from here.

However, if we looked at the equation (or had good physical intuition) we would have found that our choices are constrained. Namely, the equations tell us that the material property $\alpha = k/\rho c$ is the important one. If we started over using α in our table we would have only one choice as we remove each independent dimension.

$T - T_c$ $[\theta]$	x [L]	t [T]	$T_h - T_c$ $[\theta]$	ℓ $[L]$	α $[\frac{L^2}{T}]$		
$T - T_c$ $[\theta]$	$\frac{x}{\ell}$	t [T]	$T_h - T_c$ $[\theta]$		$\frac{\alpha}{\ell^2}$ $[\frac{1}{T}]$		remove L
$T - T_c$ $[\theta]$	$\frac{x}{\ell}$	$\frac{t\alpha}{\ell^2}$	$T_h - T_c$ $[\theta]$				remove T
$\frac{T-T_c}{T_h-T_c}$	$\frac{x}{\ell}$	$\frac{t\alpha}{\ell^2}$					remove θ

The Pi Theorem, and using the fact that the equation tells us that

α matters rather than the three parameters independently yields the dimensionless temperature is a function of the dimensionless spatial coordinate and the dimensionless time,

$$\frac{T - T_c}{T_h - T_c} = f\left(\frac{x}{\ell}, \frac{t\alpha}{\ell^2}\right).$$

The important point is that stating the equations and boundary conditions for a problem mathematically allows us to analyze the problem through dimensional analysis in a way that does not always require the physical intuition of Lord Rayleigh. The Pi Theorem also tells us that we can write the governing equation as that of a dimensionless temperature as a function of dimensionless time and dimensionless space. Dimensionless numbers time and space coordinates should bother you no more than the arbitrary definitions of the meter and the second.

5.3.2 Scaling the equation

The equivalent approach is to explicitly take the governing equations and convert the units from man made ones to dimensionless ones. We did this for a few simple problems in our first chapter on dimensional analysis. To proceed, we introduce a non-dimensional spatial coordinate $\tilde{x} = x/\ell$ so that the domain extends from $0 < \tilde{x} < 1$. We can also define a non-dimensional time as $\tilde{t} = t/t_0$ where t_0 is an arbitrary (for now) time scale. Under this change of variables, the heat equation becomes,

$$\frac{\partial T}{\partial(\tilde{t}\ t_0)} = \alpha \frac{\partial^2 T}{\partial(\ell\tilde{x})^2}. \tag{5.17}$$

Since ℓ and t_0 are constant we can pull it outside the derivatives to obtain

$$\frac{\partial T}{\partial \tilde{t}} = \frac{\alpha t_0}{\ell^2} \frac{\partial^2 T}{\partial \tilde{x}^2}. \tag{5.18}$$

Nothing is stopping us from arbitrarily setting $t_0 = \ell^2/\alpha$ so that the diffusion equation becomes simplified with no material or geometric parameters,

$$\frac{\partial T}{\partial \tilde{t}} = \frac{\partial^2 T}{\partial \tilde{x}^2}. \tag{5.19}$$

We can also scale the temperature as appropriate for the boundary

conditions. We can introduce the function,

$$\Theta = \frac{T - T_c}{T_h - T_c}. \tag{5.20}$$

Think about this change of variables for the temperature for a minute, it has a really nice feature of bounding the temperature in problem between 0 and 1. This scaling changes the boundary conditions to

$$\begin{aligned} \Theta(\tilde{x}, \tilde{t} = 0) &= 1; \\ \Theta(\tilde{x} = 0, \tilde{t} > 0) = 0; \qquad \Theta(\tilde{x} &= 1, \tilde{t} > 0) = 0. \end{aligned} \tag{5.21}$$

The scaling for temperature does not effect the governing partial differential equation, i.e.

$$\frac{\partial \Theta}{\partial \tilde{t}} = \frac{\partial^2 \Theta}{\partial \tilde{x}^2}. \tag{5.22}$$

The dimensionless equation and boundary conditions show that the solution has to have a form

$$\Theta = f(\tilde{x}, \tilde{t}) \quad \text{or} \quad \frac{T - T_c}{T_h - T_c} = f\left(\frac{x}{\ell}, \frac{t\alpha}{\ell^2}\right)$$

The functional form is exactly that obtained via the Pi Theorem.

While it may not seem so, these results are very powerful and useful; it says all diffusion problems (with this boundary condition) are the same. We simply scale the geometry to have a length that ranges from zero to one. The combination of the size and the thermal diffusivity gives us the appropriate time unit. On the scaled domain and in the proper time units, problems of different size and material property will have the same solution. Regardless of the size, material or temperatures involved, if you solve the diffusion equation for this problem you have solved it once and for all. Once we have our solution we simply use the definitions above to move from the dimensionless solution to physical units by multiplying by the appropriate constants.

Suppose we have a slab of copper ($\alpha = 1.17\text{cm}^2/\text{s}$) that is one centimeter thick, the appropriate time scale is $t_0 = \ell^2/\alpha = 1^2/1.17$ seconds. If we then double the thickness to 2 centimeters the time scale changes to $t_0 = 2^2/1.17$ seconds. This means that temperature field will be the same in the second case at a physical time that is four times longer. If you are cooking two steaks and one is twice as thick as the other, the thicker one will need to cook four times longer to achieve the

same center temperature. The scaling also shows that the cooling of an object from 200 C to 25 C is the same as cooling from 30 C to 25 C. This result is truly remarkable!

This scaling for time in the heat conduction equation is very important. It provides us with an estimate for how long things take to heat and cool. If we know the material and the length scale, then thermal equilibrium will be reached on a time scale of the order of ℓ^2/α. This is not a precise estimate, but a useful one nonetheless. The estimate tells us the order of magnitude. If the estimate tells us the time scale is one minute, we can't be sure if it will take 1 minute or 3 minutes. However, we can be confident that heating will not occur in one second or take hours. It is an order of magnitude estimate - that's all. These types of estimates based on arguments of units are useful because they are so simple and can give some immediate intuition.

5.4 General derivation of the heat equation

Now that we have worked through the derivation of the heat equation in a 1D problem, let's generalize the derivation to three dimensions. While we could proceed with the derivation based on little elements analogous to what we have already done, I am going to switch to a notation which relies heavily on vector calculus notation. While we could derive the equation without relying on vector calculus so much, the use of vector calculus will serve us well later, so it is beneficial to get used to the notation. In the general derivation we have to move to using the heat flux vector, $\mathbf{q}$ W/m^2, in 3D space.

5.4.1 Storage of thermal energy

If the temperature changes as function of position within a 3D volume, the storage of thermal energy we discussed in the 1D example is written in general integral form as,

$$\Delta E = \int \rho c(T - T_{ref}) dV. \tag{5.23}$$

The integral is taken over the entire volume of interest and dV denotes a small unit of volume. Note that before we wrote $T(x,t)$ to be explicit,

for compactness we will often just write T and we must remember that temperature is a function of time and space.

5.4.2 Heat flow across a surface

In three dimensions, we now must consider the heat flux to be a vector quantity. The rate that thermal energy flows across an arbitrary boundary is given by integrating the heat flux vector over the surface. If we take the surface of an arbitrary volume, and draw a normal vector $\mathbf{n}$ to the surface that points outward, then $\mathbf{q} \cdot \mathbf{n}$ is the rate that energy crosses the surface at that point. That the normal vector points outward is an arbitrary choice, but it is one that is commonly accepted by a "gentleman's agreement" (Schey (1997)). The dot product with the surface normal vector is important since the heat flux is a vector. If the heat flux is pointing tangential to a surface, then no heat is crossing the surface. If we consider our arbitrary surface, S, which encloses our volume the net rate that energy crosses that surface is,

$$\dot{E}_{\text{in}} - \dot{E}_{\text{out}} = -\oint \mathbf{q} \cdot \mathbf{n} dS. \tag{5.24}$$

Here the dS in the integral denotes the integral is taken across the surface, S. I have included the circle on the integral here to remind us for now that the integral is taken around the closed surface which bounds our material volume of interest. The fact that the heat flux vector has direction takes care of heat flowing in or out of the volume. The minus sign is there because the convention is to define the normal vector to point outwards from a closed surface. Evaluating $\mathbf{q} \cdot \mathbf{n}$ gives a positive result when heat is flowing out of the volume. When heat is flowing in, $\mathbf{q} \cdot \mathbf{n}$ is negative.

5.4.3 Conservation of thermal energy

If we now consider a closed volume, conservation of energy in rate form (again, only considering thermal energy and no mechanical work and no energy generation) would state,

$$\frac{d}{dt} \int \rho c (T - T_{ref}) dV = -\int \mathbf{q} \cdot \mathbf{n} dS.$$

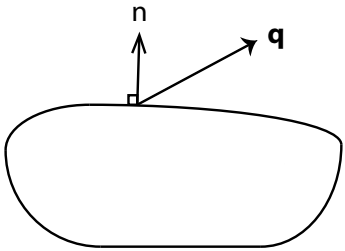


Figure 5.4 Schematic of heat flux and normal vector on an arbitrary, closed surface.

Since the reference temperature is a constant, that term goes away when we take the time derivative, thus we are left with,

$$\frac{d}{dt}\int \rho c T dV = -\int \mathbf{q}\cdot\mathbf{n} dS. \tag{5.25}$$

Physically the equation says the rate of change of the total energy stored inside our volume is equal to the net rate that heat flows across the bounding surface. The equation is exactly as we had in the one-dimensional case. This integral form of the conservation of energy will be useful to us in a number of problems, for it must always hold true for thermal problems. Note that we have dropped the circle from the integral to denote a closed surface. Since we are equating a volume and surface integral it is implied that the surface integral is over the closed surface which bounds the volume. I find the notation simpler without all the circles on the integrals.

In this chapter we consider heat conduction in a solid where the volume of integration remains fixed and does not move. The shape and mass of the solid inside the region of integration is not changing. This is a subtle point that will be important when we discuss fluid flow, because when things are moving the volume of integration may itself be changing with time. If the volume is fixed then on the left hand side the order of the derivative and integral can be switched. Further, if we assume constant material properties for ρ and c, we obtain,

$$\int \rho c \frac{\partial T}{\partial t} dV = -\int \mathbf{q}\cdot\mathbf{n} dS.$$

Note that the time derivative of the temperature is now properly writ-

ten as the partial derivative. In the previous equation, we were interested in the time derivative of the *integral* of the temperature, thus we already removed spatial dependence via integration and thus were fine to use the d/dt notation.

Finally this integral equation can be written in differential form if we make use of a little vector calculus. Remember the divergence theorem? Applying the divergence theorem to the right hand side converts the surface integral to a volume integral.

$$\int \rho c \frac{\partial T}{\partial t} dV = -\int \nabla \cdot \mathbf{q} dV.$$

We can then group everything together as

$$\int (\rho c \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q}) dV = 0.$$

The volume of integration is arbitrary, I could make it bigger, smaller or move it. The only way such an integral can *always* be zero is if the thing we are integrating is zero. Our integral equation can be written in differential form as

$$\rho c \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = 0. \tag{5.26}$$

This equation is valid at every point in our material. If you go back and review the way in which we derived the heat equation in one dimension without resorting to vector calculus notation, you will see the development is identical.

The physical interpretation of the equation is simple. If all the heat flux vectors point toward a single location then energy is flowing toward that point and we expect the temperature to increase with time. Mathematically, when the heat flux vectors point toward a single location, then the divergence of the heat flux is negative and thus our equation says that the rate of change of temperature at that location is positive. If all the heat flux vectors point away from a point, then the temperature at that location decreases with respect to time. In one dimension the heat flux had to be constant for us to have an unchanging temperature field. In three dimensions the heat flux field has to have zero divergence for there to be no change in temperature. At a point, the heat flux vectors could be converging in the x-direction but diverging in the y direction, thus local amount of energy flowing to that point

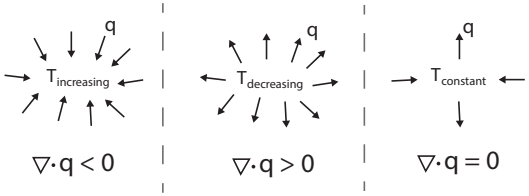


Figure 5.5 Schematic of the divergence of the heat flux. The temperature at a point rises, falls, or stays the same depending on the local divergence of the heat flux vectors.

on net zero and thus the temperature does not change. The regimes of positive, negative, and zero divergence are shown in Figure 5.5.

The number of assumptions we used to arrive at this point are minimal. We are only considering thermal energy in a solid with no thermal energy generation due to chemical/nuclear reactions inside the volume. There is no mechanical work and the solid is not deforming.

5.4.4 Fourier's law

So our equation is beautiful to look at,

$$\rho c \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = 0,$$

but it is not complete. There are four unknowns, 3 components of the heat flux vector and temperature, and only one equation. Closure to the problem is provided via Fourier's law as it was in the 1D case. Fourier's law is an example of a constitutive law. Constitutive laws capture the microscopic material behavior of our continuum. The use of a constitutive law is always needed to close conservation laws. The conservation law is based on a fundamental (but unprovable) assertion that energy is conserved. We will then require a constitutive relationship which characterizes the material to complete the problem.

Generalized Fourier's law states,

$$\mathbf{q} = -k \nabla T. \tag{5.27}$$

There is a linear relationship between heat flux and temperature gradient. Remember that temperature is a scalar field. It has a single value

at each location in space. The gradient operator creates a vector from the scalar field. The negative sign in the equation is there so that heat flux will point from hot to cold.

5.4.5 Heat equation

The use of Fourier's law in our conservation equation yields,

$$\rho c \frac{\partial T}{\partial t} = \nabla \cdot (k \nabla T). \tag{5.28}$$

This equation is known as the heat equation. It is a single equation for a single unknown, the temperature. On the right hand side, the gradient of the temperature creates a vector. The divergence of that vector creates a scalar. The left hand side is a scalar (we only see T), so this is good - scalar equals scalar. When the thermal conductivity is a constant, which is often but not always the case, our equation simplifies to,

$$\frac{\partial T}{\partial t} = \frac{k}{\rho c} \nabla^2 T. \tag{5.29}$$

This form is nice because the dependence on two material properties, ρc, and k has been reduced to one. The thermal diffusivity $\alpha = k/\rho c$ has units of m^2/s.

At steady state when the temperature is not changing with respect to time, the temperature field must have zero curvature. The temperature field can be curved in the x-direction one way as long as it is curved equal and opposite in the y direction.

5.4.6 Boundary conditions

In order to solve the heat equation, we must describe what occurs at the boundary of our domain. In a one-dimensional problem this would be, for example, the temperature at the two end points of the domain. In three dimensions we need a condition along the entire surface bounding the volume where we solve the equation. Thinking physically for a moment, the boundary conditions express how the object of interest is connected to the rest of the universe. My equation is good for heat flow in the object of interest, but I can subject the object to many different environments; hot, cold, submerged in water, sitting on the

countertop in air, etc. The boundary conditions are the mathematical expression of what is happening *outside* the object where we are solving the equation.

There are some common classes of boundary conditions. The first is **fixed temperature**. This condition simply means that we know and prescribe the temperature along the boundary. In one-dimension we might write, for example, $T(t, x = 0) = 500$ K. The second condition is **insulating**. Here, we set the heat flux through the surface to be zero $\mathbf{q} \cdot \mathbf{n} = 0$. In terms of the temperature, Fourier's law says the insulating condition is $\nabla T \cdot \mathbf{n} = 0$. In one dimension this might be, for example, $\partial T / \partial x|_{x=0} = 0$. The insulating condition means that the temperature field has no slope at the boundary. We could also **prescribe the heat flux** at the boundary, which would be stated as $-k \nabla T \cdot \mathbf{n} = q$.

Another common boundary condition is to have **convection** at a surface. This boundary condition states the heat flux at a surface by conduction, must equal that of convection to a surrounding fluid. This condition would be used if modeling something cooling in air on the table. Mathematically, the convection condition is stated as $-k \nabla T \cdot \mathbf{n} = h(T - T_\infty)$. The final condition is **radiation** at a surface, which we don't consider in this course but is important in a number of problems.

The heat equation is a quite accurate description of heat flow in solids. Most of the uncertainty or error in a calculation will come through the boundary conditions. For example, in convection problems the magical convection coefficient, h, is rarely known with much accuracy.

6

Solutions to the heat equation

The heat equation,

$$\frac{\partial T}{\partial t} = \alpha \nabla^2 T, \tag{6.1}$$

with appropriate boundary and initial conditions can provide the temperature as a function of time and all three spatial dimensions in a solid (or stationary fluid). The equation is linear and well-behaved such that we can usually obtain a solution to the problem of interest. If we make some simplifying assumptions, we can often solve the problem analytically. By analytically, I mean I can write an equation on a piece of paper which is the solution to the equation and satisfies the boundary condition.

Usually a simplifying assumption means that we are removing dimensions of time or space. For example, only considering heat flow in one dimension, or assuming that the problem has reached a steady state equilibrium and no longer evolving in time. The simplest case is 1D steady state behavior which is extremely easy to calculate. While this assumption might seem limited, such an analysis often allows us to make simple estimates and obtain approximate results quickly. In one dimension, even with transient behavior we can often solve the problem analytically. We can also make some progress analytically in higher dimensions in simple geometries with simple boundary conditions.

In complicated geometries or with complicated boundary conditions we can use computer simulation - usually finite element software to solve the problem numerically. There are a number of free and commercial codes that can be used to solve such problems. Numerically, the

heat equation is generally a well behaved and well understood equation, thus numerical solutions are usually (but not always) obtained without much difficulty with modern software.

While simulation can be very useful in solving the heat equation, some of the ideas and techniques in this chapter are still extremely useful. Don't just run to the computer. Simple analytical solutions that are easy to evaluate, provide the parameters that dominate a problem, or collapse all the solutions to a single plot or equation are really handy to have. There is both beauty and utility in simplicity.

Even though the solution to the conduction problem via simulation can be very accurate for the heat diffusion problem at hand, there is often a lot of uncertainty from the boundary conditions. The uncertainty in the boundary condition means that if we compared the simulation to experiment for a realistic problem, we might have poor agreement even though the numerical solution itself is very accurate. A more accurate solution to the *conduction* problem does not mean that the *overall* solution is better than a simple approach.

6.1 1D transient behavior

Transient behavior in one spatial dimension can usually be solved by analytical means. We will not make these methods a focus of this course, however, Appendix A provides the details in solving the first problem of quenching as an example. You should at least skim this section such that you can see how this equation is solved. However, below we will focus on the behavior of the solution rather than how to obtain it.

6.1.1 Quenching

Let's consider the transient behavior of the quenching problem that we stated earlier, a hot 1D object of length ℓ suddenly cooled at the boundaries. Mathematically the problem is stated as,

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \tag{6.2}$$

$$T(x, t = 0) = T_h;$$

$$T(x = 0, t > 0) = T_c; \qquad T(x = \ell, t > 0) = T_c. \tag{6.3}$$

Read the above equations carefully. Don't just scan them. Spend some time to make sure that the equations "speak" to you. These equations are encoding all the physical understanding we have about this problem. Pay careful attention when we state a problem that we have properly defined the equation, boundary conditions, and initial condition.

In the previous chapter, we worked through how this problem could be stated in dimensionless form as,

$$\begin{aligned}
&\frac{\partial \Theta}{\partial \tilde{t}} = \frac{\partial^2 \Theta}{\partial \tilde{x}^2}, \\
&\Theta(\tilde{x}, \tilde{t} = 0) = 1; \\
&\Theta(\tilde{x} = 0, \tilde{t} > 0) = 0; \qquad \Theta(\tilde{x} = 1, \tilde{t} > 0) = 0,
\end{aligned}$$

where $\Theta = (T - T_c)/(T_h - T_c)$, x is made dimensionless by ℓ, and time is made dimensionless by ℓ^2/α. There are no free parameters, they have all been scaled out of the problem. In Figure 6.1, we show solutions to this equation, showing the spatial temperature profile at a few select instances in time. Remember that this figure is good for all such quenching problems, regardless of the material, domain size, or temperatures involved. To convert the dimensionless temperature to the real temperature, just multiply by $T_h - T_c$ and add T_c. To convert the time to seconds, just multiply time by ℓ^2/α. Note that given the time scaling a material twice as thick will take four times longer to cool.

The equation is readily solved and the solution for this problem is found in Appendix A. The analytical solution for this problem on the domain $0 < \tilde{x} < 1$ is,

$$\Theta(\tilde{x}, \tilde{t}) = \sum_{n=1,3,5\ldots}^{\infty} \frac{4}{n\pi} \mathrm{e}^{-n^2\pi^2\tilde{t}} \sin(n\pi\tilde{x}),$$

or in dimensional terms on the domain $0 < x < \ell$,

$$T(x,t) = T_c + (T_h - T_c) \sum_{n=1,3,5\ldots}^{\infty} \frac{4}{n\pi} \mathrm{e}^{-n^2\pi^2\alpha t/\ell^2} \sin\left(\frac{n\pi x}{\ell}\right).$$

Note that the solution is given as an infinite sum of sines, thus even though the solution is an exact analytical expression that we still need the compute to visualize the solution.

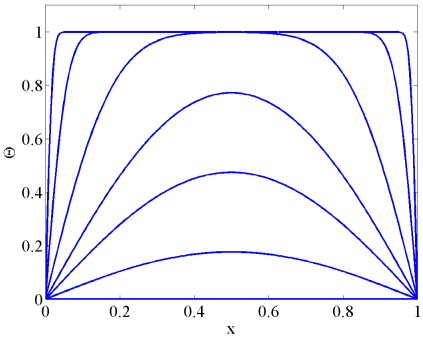


Figure 6.1 Solution to heat equation for the quenching problem. Solutions shown at dimensionless times of t =0.0001, 0.001, 0.01, 0.05, 0.1, and 0.2.

6.1.2 Quenching by convection

Now let's consider a quenching problem where a hot 1D solid is placed in a fluid and the ends are cooled by *convection*. We will assume that the object starts at T_h and is immersed in a fluid with temperature T_∞ and known convection coefficient, h. In this case, the problem can be stated in dimensional terms as,

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2},$$
$$T(x, t = 0) = T_h;$$
$$k\frac{\partial T}{\partial x}\bigg|_{x=0} = h(T - T_\infty); \qquad k\frac{\partial T}{\partial x}\bigg|_{x=\ell} = -h(T - T_\infty).$$

The boundary condition is physically stating that the heat flux by conduction right at the edge of the material equals the heat flux leaving the object by convection. These two heat fluxes must be equal right at he surface as the surface cannot accumulate thermal energy. The difference in sign on the boundary condition has to do with the direction the heat is flowing. On the left boundary $x = 0$, imagine the body is hot and the fluid is cold. The temperature gradient at the wall should be positive. On the right boundary, the temperature gradient at the wall should be negative. Note that the expression for the boundary condition says that it is the ratio h/k that matters (not h on its own).

Let's quickly see what the Pi theorem would tell us about the problem. In dimensional terms, $T = f(T_h, T_\infty, \ell, t, \alpha, h/k)$ From our previous analysis, we have seen that it is always useful to recast the temperature as

$$\Theta = \frac{T - T_\infty}{T_h - T_\infty}$$

Such as temperature ratio function is useful because it starts with a value of 1 at $t = 0$ and ends at a value of 0 when the object has equilibrated. Performing the dimensional analysis,

$\frac{T-T_\in}{T_h-T_\infty}$	x	[L]	t	[T]	ℓ [L]	α	[$\frac{L^2}{T}$]	$\frac{h}{k}$	[$\frac{1}{L}$]	
$\frac{T-T_\infty}{T_h-T_\infty}$	$\frac{x}{\ell}$		t	[T]		$\frac{\alpha}{\ell^2}$	[$\frac{1}{T}$]	$\frac{h\ell}{k}$		Remove L
$\frac{T-T_\infty}{T_h-T_\infty}$	$\frac{x}{\ell}$					$\frac{\alpha t}{\ell^2}$		$\frac{h\ell}{k}$		Remove T

The dimensionless result is

$$\Theta = f\left(\frac{x}{\ell}, \frac{\alpha t}{\ell^2}, \frac{h\ell}{k}\right).$$

We can use the equations or the table to arrive at the "right" dimensionless parameters, let's redo the analysis from the perspective of the equations. We can make the equation dimensionless as before defining, $\tilde{x} = x/\ell$ and $\tilde{t} = t\alpha/\ell^2$. The question is how to scale the boundary conditions. The scaling at this point is precisely as in the previous problem. In dimensionless terms our problem is stated as,

$$\begin{aligned}
&\frac{\partial \Theta}{\partial \tilde{t}} = \frac{\partial^2 \Theta}{\partial \tilde{x}^2}, \\
&\Theta(t = 0, x) = 1; \\
&\left.\frac{\partial \Theta}{\partial \tilde{x}}\right|_{x=0} = \frac{h\ell}{k}\Theta; \qquad \left.\frac{\partial \Theta}{\partial \tilde{x}}\right|_{x=1} = -\frac{h\ell}{k}\Theta.
\end{aligned} \tag{6.4}$$

Whether we prefer to work from the equations or the table, either way the analysis introduces a new parameter, $\frac{h\ell}{k}$, which is known as the Biot number. The Biot number characterizes the ratio of the resistance to heat transfer by conduction through the body to the resistance to heat transfer by convection. If the Biot number is Bi >> 1, then the conduction resistance is the largest. Convection is vigorous. In this case would expect the boundaries to be very close to the fluid temperature

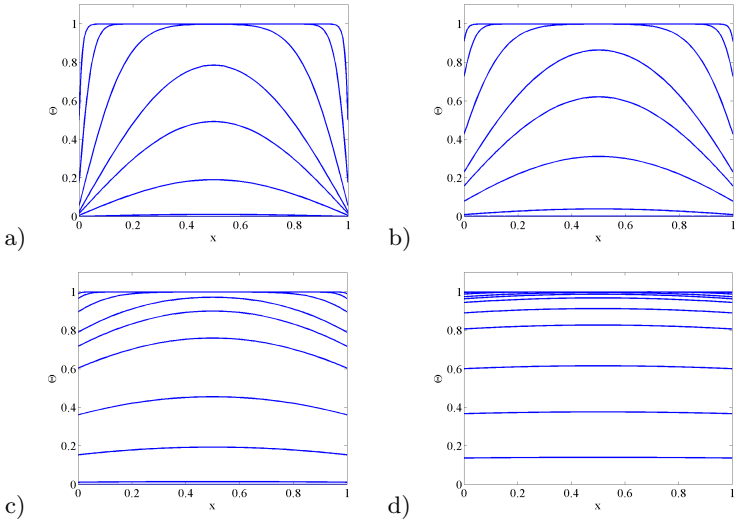


Figure 6.2 Solution to heat equation for the quenching problem. Solutions shown at dimensionless times of t =0.0001, 0.001, 0.01, 0.05, 0.1, 0.2, 0.5, 1, 2.5, 5, and 10. The Biot numbers are a) 100, b) 10, c) 1, and d) 0.1.

$\Theta(x = 0) \approx 0$ and the problem approximates that of the previous section. In the limit of Bi << 1, then the convection resistance is large. In this case, we can assume that the time it takes heat to conduct from the center to wall is so fast that the temperature of the body is approximately constant in space. The body just cools at nearly a uniform temperature.

Solutions to the equation for different Biot numbers are shown in Figure 6.2. At Bi = 100 the solutions look just like the solution of the previous section. At Bi = 0.1 the solutions show very minor spatial temperature gradients. At low Biot numbers then, we could ignore the spatial distribution of temperature and treat the body as having a single temperature that is only a function of time.

6.1.3 Quenching when Bi << 1: Lumped model

Let's now consider a simplification that is useful in many instances. Let's start with the governing equation,

$$\rho c \frac{\partial T}{\partial t} = k \frac{\partial^2 T}{\partial x^2}.$$

Since we are looking to get rid of the spatial component of T, let's integrate the entire equation across the domain,

$$\int_0^\ell \left(\rho c \frac{\partial T}{\partial t} = k \frac{\partial^2 T}{\partial x^2} dx \right).$$

Since the region of integration is fixed, we can interchange the order of differentiation and integration on the left hand side. On the right, the integral of a derivative is easy to perform. The result is,

$$\rho c \frac{d}{dt} \int_0^\ell T dx = k \left. \frac{\partial T}{\partial x} \right|_{x=\ell} - k \left. \frac{\partial T}{\partial x} \right|_{x=0}.$$

Let's define the object's mean temperature as $\bar{T}(t) = \frac{1}{\ell} \int_0^\ell T dx$ and by using symmetry at the two boundaries,

$$\rho c \ell \frac{d\bar{T}}{dt} = -2k \left. \frac{\partial T}{\partial x} \right|_{x=0}.$$

Substituting the convection boundary condition $k \left. \frac{\partial T}{\partial x} \right|_{x=0} = h(T(0,t) - T_\infty)$ yields,

$$\rho c \ell \frac{d\bar{T}(t)}{dt} = -2h(T(0,t) - T_\infty). \tag{6.5}$$

This expression for the rate of change of the mean temperature is always true for this problem - we have not yet made any assumptions.

In Figure 6.2 we saw if the Biot number was small the temperature in the object is realtively constant. Under the approximation of uniform spatial temperature, the boundary temperature and the mean temperature are about the same. In this limit we would obtain an ordinary differential equation for the object's one and only temperature, $T(t)$,

$$\frac{d\ T(t)}{dt} = -\frac{2h}{\rho c \ell}(T(t) - T_\infty). \tag{6.6}$$

The model gives a single parameter $\rho c \ell / 2h$ which has units of time. Therefore, from a dimensional anlaysis perspective, this new time scale

is the only parameter in town. Therefore, time scale for cooling is set by $t_0 = \rho c\ell/2h$. Note that the cooling time scale does not depend on the material itself, which might at first seem counterintuitive. Once a material is conductive enough such that its temperature is approximately constant, making it more conductive doesn't change the resulting behavior.

We can proceed by making the model completely dimensionless. If we take the cooling equation and define a dimensionless temperature as $\Theta = (T - T_\infty)/(T_h - T_\infty)$ and an arbitrary time scale $t = t_0\tilde{t}$, then we have

$$\frac{d\Theta}{d\tilde{t}} = -\frac{2ht_0}{\rho c\ell}\Theta.$$

We are led to the conclusion that the natural unit of time is $t_0 = \frac{\rho c\ell}{2h}$, such that we obtain,

$$\frac{d\Theta}{d\tilde{t}} = -\Theta,$$

with initial condition $\Theta(0) = 1$. The solution to the above dimensionless equation is an exponential for the object's cooling,

$$\Theta = \mathrm{e}^{-\tilde{t}}$$

The dimensionless formulation shows us that under the lumped assumption, all objects cool along the same *dimensionless* curve. An example of some experimental data collapsed using the analysis presented here is shown in Figure 6.3.

6.1.4 Interpretation of the Biot number

In Figure 6.2 we showed the Biot number provided a good estimate for when the object could be considered at a uniform temperature. A little physical reasoning can verify that the Biot number is critical to understanding if the lumped model is a good one. Let's see what a little simple estimation can do for us. Let's define the internal temperature difference ΔT_{int} as the temperature difference from the objects center to the boundary. At any instant, the heat flux leaving the object by conduction is *estimated* by applying Fourier's law from the center to the boundary,

$$q_{\text{cond}} \approx \frac{2k}{\ell}\Delta T_{\text{int}}.$$

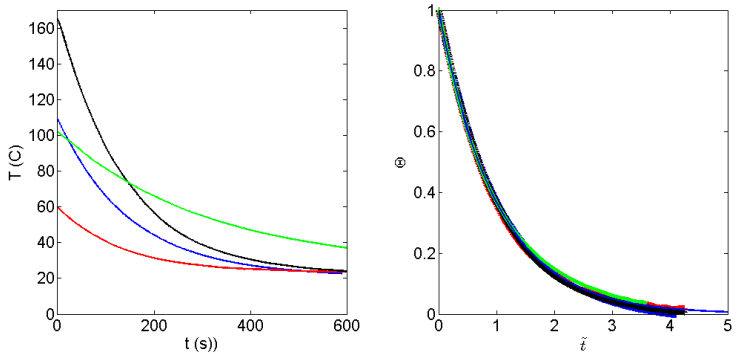


Figure 6.3 Experimental data taken by the author for 4 objects heated to different temperatures and subjected to different conditions. On the right are the data in dimensional form, on the left are the data collapsed by the scaling suggested in the analysis. The experiments are compare to $\Theta = \mathrm{e}^{-t}$, the solution to the lumped model.

Let's define the external temperature difference ΔT_{ext} as the temperature difference between the objects boundary and the fluid. The heat flux at the boundary by convection is given as,

$$q_{\text{conv}} = h\Delta T_{\text{ext}}.$$

Since these two heat fluxes must be equal (though the flux by conduction is only an estimate),

$$\frac{2k}{\ell}\Delta T_{\text{int}} \approx h\Delta T_{\text{ext}}$$

which is rearranged to give,

$$\frac{\Delta T_{\text{int}}}{\Delta T_{\text{ext}}} =\approx \frac{h\ell}{2k}$$

In the limit of small Biot number, $Bi = \frac{h\ell}{k} << 1$, then the temperature difference inside the object is small relative to the temperature difference between the object and it's surroundings; the uniform temperature approximation is valid.

We can also think about the Biot number in terms of time scales. The general problem of quenching by convection has two thermal time scales. One scale is how long it takes for heat to get from the center

of our object out to the edge. This time scale is the conduction time, $\ell^2/\alpha = \ell^2\rho c/k$. The other is how long it takes heat to leave the solid and be carried away by the fluid which is $\rho c\ell/2h$. The ratio of these two time scales is $\frac{\ell^2\rho c}{k}\frac{2h}{\rho c\ell} = \frac{2h\ell}{k}$, or the Biot number! When the Biot number is small it takes a very short time for heat to get from the center of the solid to the boundary relative to how long it takes heat to make its way into the fluid. Thus in the limit of small Biot number changes inside the solid equilibrate "instantly" compared to the rate that overall temperature is dropping. In the other limit of large Biot number, it means that it takes a long time for heat to make its way from the center of the solid to the boundary. In this limit we expect the boundary to reach the fluid temperature.

When approached with a quenching problem, even in complicated geometry, the first thing to do is estimate the Biot number. The Biot number provides a reliable and simple estimate about the basic physical picture. We can think of the Biot number as either a ratio of time scales or a ratio of thermal resistances. Both interpretations are equally valid.

6.2 1D Steady state

Steady state refers to when a system has reached equilibrium and is no longer changing with respect to time. In one dimension, the steady state problem says that,

$$\frac{\partial^2 T}{\partial x^2} = 0.$$

The general solution to this problem is

$$T(x) = ax + b,$$

where a and b are constants which are determined from the two boundary conditions. In terms of the heat flux, when the temperature is linear the heat flux is constant everywhere.

If we consider a domain $0 < x < \ell$ with boundary conditions of holding the temperature at the two ends fixed, we obtain,

$$T(x) = (T(0) - T(\ell))\frac{\ell - x}{\ell} + T(L).$$

The heat flux, obtained from $\mathbf{q} = -k\nabla T$ is,

$$q = k\frac{T(0) - T(\ell)}{\ell} = \frac{k}{\ell}\Delta T.$$

If we think of ℓ/k as the thermal resistance, then we have a thermal equivalent to Ohm's law where the temperature difference is analogous to the voltage drop and the heat flux is analogous to the current.

$$\Delta T = qR \quad \text{where} \quad R = \frac{\ell}{k}.$$

This is a simple and remarkably useful result. It is useful for obtaining simple estimates of heat flows. Take a building in the winter; cold on the outside and hot on the inside. The above equation can be used to estimate losses through the walls and windows. You may have heard of R-values for insulation. The R-value is essentially the value of ℓ/k. In practice R-values in the US are given in English units and sometimes there is a factor which includes the area and sometimes they are given per unit area. Real calculations with R-factors are conceptually pretty simple but the use of different units in industry makes the reality somewhat more confusing.

6.2.1 Resistor networks - 1D steady state heat flow

The analogy to resistor circuits extends beyond the simple case above. Two materials in series with a temperature drop across both can be modeled as two resistors in series. Two materials side by side with the same temperature difference imposed across both can be modeled as resistors in parallel. However, we need to be careful as we move to resistors in parallel because the heat flux is a per unit area quantity, thus if the two materials in parallel have different areas we need to be careful. The heat flux through a material with a temperature gradient is,

$$q = \frac{k}{\ell}\Delta T. \tag{6.7}$$

The total heat flow, Q, in Watts is

$$Q = \frac{kA}{\ell}\Delta T, \tag{6.8}$$

where A is the area through which the heat flux passes. Thus, when working with resistor analogies we need to be a little bit careful if we are

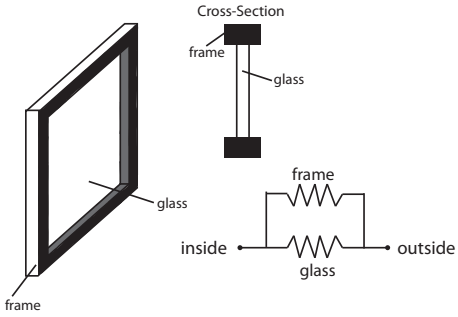


Figure 6.4 Window glass surrounded by a metal frame. The resistance to heat flow through the window and the frame act as resistors in parallel.

talking about total heat flow or heat flux. When working with resistor networks we will usually include the area to keep things clear and thus we use,

$$\Delta T = QR \tag{6.9}$$

where the thermal resistance is given as $R = \ell/kA$.

A composite wall can be modeled as a resistor network. Again, this simple model is limited. It assumes all heat flow is one dimensional and the system is at steady state. However, it is very powerful technique for estimating heat flows in systems. For example, in my office I have a large window with a metal frame around the outside. In the winter, the metal frame on the inside of my office gets extremely cold. The metal frame and the window (which is double-paned glass) are like resistors in parallel. The metal frame allows a significant amount of heat flow around the window, just like a small resistor "shorting out" a large one, shown in Figure 6.4 All we need to do is measure/know k, ℓ, and A and we can make a good estimate of whether the load due to the window frame is significant. While these resistor network solutions are not always rigorous solutions to the heat heat equation, they are immensely valuable. The resistor analogy provides a very simple way to estimate the magnitudes of heat flow and do quick calculations.

You should also realize that in problems dominated by convection effects, the resistor analysis is perfectly acceptable. Rarely are values

of h known precisely nor are they ever constant in time or location on the surface. Errors in conduction analysis are often overwhelmed by the difficulty of knowing the convection coefficient with any accuracy.

6.2.2 Resistor analysis example

Imagine a warm room separated by a wall to the cold outdoors. Heat flows continuously from inside to out through the wall. At steady state, there are three processes by which heat moves from inside to out; convection from the air to the surface of the wall indoors, conduction through the wall, and convection from the surface of the wall to the air outdoors.

For heat flow by conduction through the wall, we can use the resistor analogy,

$$q_2 = \frac{k}{\ell}\Delta T = \frac{k}{\ell}(T_1 - T_2), \tag{6.10}$$

where the subscript w refers to the wall temperature inside and out. For convection at the indoor surface of the wall we have,

$$q_1 = h_{in}(T_{in} - T_1), \tag{6.11}$$

which follows the same resistor analogy where the resistance due to convection is $R = \frac{1}{h}$. Likewise, for the exterior of the wall we have heat flow by convection,

$$q_3 = h_{out}(T_2 - T_{out}). \tag{6.12}$$

Since the heat flow from the internal air to the wall must equal the heat flow through the wall and equal to the heat flow from the wall to the exterior air, we have

$$q_1 = q_2 = q_3 = q$$

At steady state, all the heat flows in this example are equal and there is a single q. If the heat flows were not equal then energy would be accumulating and the system would not be at steady state. The problem is the equivalent to resistors in series as shown in Figure 6.5

We have three resistors in series; indoor convection, wall conduction, and outdoor convection. The total resistance is the sum of the three resistances,

$$R = \frac{1}{h_{in}} + \frac{\ell}{k} + \frac{1}{h_{out}}.$$

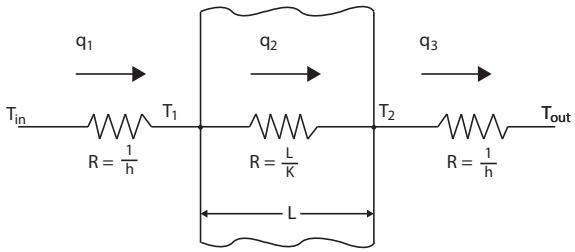


Figure 6.5 Heat flow through a 1D wall at steady state with convection on the interior and exterior of the wall. The three resistances to heat flow act in series.

The heat flux from indoors to out is then the total temperature difference divided by this resistance,

$$q = \frac{T_{in} - T_{out}}{R} = \frac{T_{in} - T_{out}}{\frac{1}{h_{in}} + \frac{\ell}{k} + \frac{1}{h_{out}}}.$$

The temperature of the wall indoors can be found in the same way we find voltages at intermediate nodes in a set of resistors in series, namely by equating the heat flow through different resistors. Here, we equate the heat flow through the whole system to the heat flow from the indoor air to the wall by convection,

$$q = \frac{T_{in} - T_{out}}{\frac{1}{h_{in}} + \frac{\ell}{k} + \frac{1}{h_{out}}} = \frac{T_{in} - T_1}{\frac{1}{h_{in}}},$$

which we can rearrange to get

$$\frac{T_{in} - T_1}{T_{in} - T_{out}} = \frac{\frac{1}{h_{in}}}{\frac{1}{h_{out}} + \frac{\ell}{k} + \frac{1}{h_{out}}}.$$

In the case (for simplicity) where $h_{in} = h_{out}$, the equation can be rewritten as

$$\frac{T_{in} - T_1}{T_{in} - T_{out}} = \frac{1}{2 + \frac{h\ell}{k}}.$$

In the limit where convection is vigorous and the wall is relatively resistive, the Biot number, $h\ell/k$, is large and

$$\frac{T_{in} - T_1}{T_{in} - T_{out}} \to 0,$$

meaning $T_1 \to T_{in}$. A thick, well insulated wall feels the same temperature as the room you are in. In the case where conduction resistance is very small compared to convection, the Biot number is small and

$$\frac{T_{in} - T_1}{T_{in} - T_{out}} \to \frac{1}{2},$$

meaning $T_1 \to (T_{in} + T_{out})/2$. If the wall is thin and poorly insulted it will be a temperature halfway between the indoor and and outdoor temperature.

Note that we were able to extract some important things with this simple analysis. First, we were able to easily get the answer for how much heat flow is through our system. Even if the geometry was more complicated such that our resistor model is not that accurate, remember that our knowledge of h is not that accurate in the first place. A more accurate geometric model doesn't mean a more accurate result overall. Second, we easily able to extract some insight into the limiting behavior to provide some insight to the physics and very little cost. We even found the single controlling dimensionless parameter. It would be harder to see these kind of trends with a computer simulation, for example.

6.3 Steady state, 2D

If a system comes to steady state and nothing is changing in time, then the heat equation says that,

$$\nabla^2 T = 0. \tag{6.13}$$

The statement is that the temperature field has no curvature. I don't know how to think about curvature in 3D. In 2D, it takes a little time to think about but it is possible to visualize. Things can looked "curved" but have no curvature in the two dimensions. A saddle is one example. In one direction it curves up in the other it curves down, thus the total curvature is zero.

An example solution to the steady state equation, known in mathematics as Laplace's equation, is shown in Figure 6.6. In this solution the wall on the left is held at a temperature of 1 and all other wall are held at zero. Look at the image and follow a line from top to bottom along

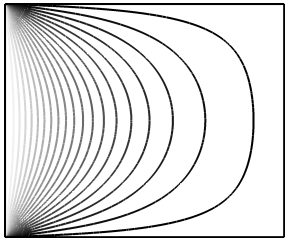


Figure 6.6 Solution to heat equation at steady state in a 2D box. The left wall is help hot while the other three walls are held cold. The contours show lines of constant temperature.

the middle of the domain. The temperature starts at zero at the upper wall, increases, then decreases again. The curvature is thus downward. This means if we follow a line from left to right, the temperature field in this direction would have curvature upwards. I encourage you to use the finite element software to visualize solutions to the heat equation in 2D at steady state.

6.4 Effective 1D heat flow

In the previous section on quenching, the lumped approximation allowed us to consider a simpler approximation by removing the spatial variation of the temperature field and only consider the dynamics. The cooling object was described by a single temperature. Another related approximation occurs in problems where heat flow occurs in 2 dimensions (or more) but the dominant temperature distribution is along one dimension.

As an example, consider heat transfer in a cylinder of radius R and length ℓ. The cylinder is attached at one end to a thermal reservoir at $x = 0$ held at temperature T_h. The surface of the cylinder at $r = R$ is cooled by convection with coefficient h and temperature T_∞. We consider only the axisymmetric case. We would like to describe the temperature variation within the cylinder. A schematic is shown in Figure 6.7. Intuitively we know that the temperature will decrease as

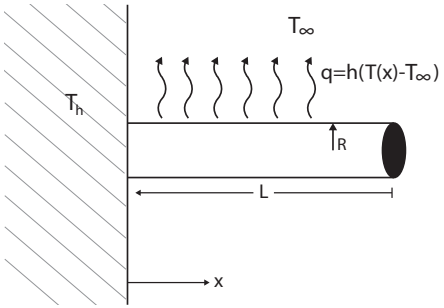


Figure 6.7 1D heat conduction through a cylinder attached to a hot wall.

we move away from the wall and if the cylinder is long and skinny the end temperature will be close to that of the cool fluid. If we wanted to know the steady state temperature we would need to solve $\nabla^2 T = 0$. The use of other coordinate systems is one reason the ∇^2 notation is nice. We always write our equations in vector notation and then for a specific coordinate system we can look up the operator in component form. In this case we will want to work in cylindrical coordinates.

The steady heat axisymmetric heat equation in cylindrical coordinates, $T(x, r)$, would be stated as,

$$k\left(\frac{\partial^2 T}{\partial x^2} + \frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right)\right) = 0$$

subject to boundary conditions at the left and right ends of the rod,

$$T(0, r) = T_h, \qquad k\frac{\partial T}{\partial x}\bigg|_{x=\ell} = h(T(\ell, r) - T_\infty)$$

as well as one along the cylindrical surface of the rod,

$$-k\frac{\partial T}{\partial r}\bigg|_{r=R} = h(T(x, R) - T_\infty)$$

on the domain

$$0 < r < R, \quad 0 < x < \ell.$$

Notice the the extra factors of r that do not appear in cartesian coordinates.

While it is possible to numerically solve this equation for the 2D

temperature distribution, let us consider the case where the cylinder radius is somewhat small relative to the length and the material is rather conductive. In this case, it seems logical to assume that the temperature distribution will mostly change in the axial direction.

As with the lumped analysis, if you want to try and construct a simpler model for your problem the first thing to try is taking an integral or average. Let's take the integral of this equation over the cross sectional area, i.e.

$$\int_0^R \left(k \left(\frac{\partial^2 T}{\partial x^2} + \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial T}{\partial r} \right) \right) = 0 \right) 2\pi r dr.$$

which is equivalent to

$$k \frac{d^2}{dx^2} \left(\int_0^R T 2\pi r dr \right) + k \int_0^R 2\pi \frac{\partial}{\partial r} \left(r \frac{\partial T}{\partial r} \right) dr = 0$$

Recognizing that the mean temperature at a given axial location would be defined as

$$\bar{T}(x) = \frac{1}{\pi R^2} \left(\int_0^R T 2\pi r dr \right)$$

we have

$$k\pi R^2 \frac{d^2 \bar{T}}{dx^2} + 2\pi k R \left. \frac{\partial T}{\partial r} \right|_{r=R} = 0$$

From the boundary condition at the outside surface of the cylinder we have,

$$kR \frac{d^2 \bar{T}}{dx^2} - 2h(T(x, R) - T_\infty) = 0$$

Finally, our equation is

$$\frac{d^2 \bar{T}}{dx^2} = \frac{2h}{kR} (T(x, R) - T_\infty).$$

So far, the above equation is always true. However, we can only really do something with it if we make an approximation. If we assume that the temperature of the cylinder only varies along the axial direction, $\bar{T}(x) \approx T(x, R)$, then temperature is a function of x only. Thus,

$$\frac{d^2 \bar{T}}{dx^2} = \frac{2h}{kR} (\bar{T} - T_\infty). \tag{6.14}$$

As with the lumped analysis, we need some criteria to let us know

when we can make the approximation that temperature only depends upon axial location? This parameter is hR/k, which is the Biot number based on the cylinder radius, When the *radial* Biot number is small then the 1D model should be appropriate. As with the lumped model in a previous section, the Biot number is representing the strength of convection relative to conduction.

6.4.1 Dimensionless form

Now let's look at the solution to the Equation 6.14 for $T(x)$. The equation we solve is

$$\frac{d^2(T-T_\infty)}{dx^2} = \frac{2h}{kR}(T-T_\infty).$$

with the conditions that

$$T(0) = T_h, \qquad k\frac{\partial T}{\partial x}\bigg|_{x=\ell} = h(T(\ell)-T_\infty),$$

on the domain, $0 < x < \ell$. Let's scale the domain $\tilde{x} = x/\ell$. We will scale the temperature by defining

$$\Theta = \frac{T-T_\infty}{T_h-T_\infty}.$$

Making the substitution we get

$$\frac{d^2\Theta}{d\tilde{x}^2} = \frac{2h\ell^2}{kR}\Theta = m^2\Theta,$$

where $m^2 = \frac{2h\ell^2}{kR}$ and the equation is subject to boundary conditions

$$\Theta(0) = 1, \qquad \frac{\partial\Theta}{\partial\tilde{x}}\bigg|_{\tilde{x}=1} = -\frac{h\ell}{k}\Theta,$$

on the domain

$$0 < \tilde{x} < 1.$$

The general solution to the equation is

$$\Theta = A\cosh(m\tilde{x}) + B\sinh(m\tilde{x}).$$

You can confirm by inspection that the solution works. Applying the boundary condition at $\tilde{x} = 0$ we get $A = 1$. Applying the boundary

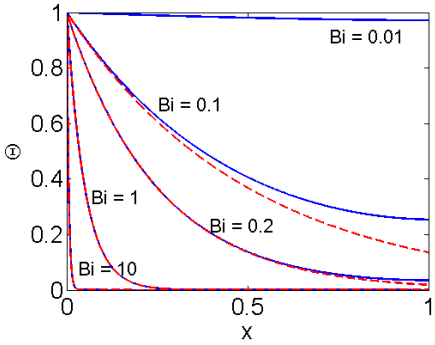


Figure 6.8 Solutions to the dimensionless problem for heat conduction in a rod where one end is held at a fixed temperature and all other surfaces are exposed to convection. For all curves the ratio of length to radius is fixed at $\ell/R = 10$, while the solid curves at for values of Biot number, $Bi = h\ell/k = 0.01,\ 0.1,\ 0.2,\ 1,\ 10$. The dashed curves show the approximation $\Theta = \exp(-mx)$; this approximation is valid for large values of m.

condition at $\tilde{x} = 1$ we get B and the final result is

$$\Theta = \cosh(m\tilde{x}) - \frac{\cosh(m) + \frac{2\ell}{R}\sinh(m)}{\sinh(m) + \frac{2\ell}{R}\cosh(m)}\sinh(m\tilde{x})$$

Some sample solutions are shown in Figure 6.8. In the limit of large values of the parameter m, the solution limits a simpler form of $\Theta = \exp(-mx)$, which is shown compared to the actual solution.

While this might all seem like a mathematical exercise, the result is important. The result represents a way to compute and predict heat transfer from a fin or extended surface. A common way to improve heat transfer is to increase the surface area. You have probably seen something similar with the fins on a car's radiator or perhaps on a heat sink for an audio amplifier. Since heat transfer to the air is often the rate limiting step, we often add extra surface area to pull more heat away. In our example above, a surface with a lot of small skinny rods attached would have better heat transfer performance than a smooth surface. This exact model of small cylinders attached to a surface has been used explain and model how the sparse hair density on elephants is used to help regulate their body temperatures, for example.

6.5 Simulations

The heat equation in any 3D geometry can readily be solved with modern simulation software. Since the heat equation is generally well behaved both commercial or open-source software solutions work without much effort or training. Many mechanical design and CAD softwares have thermal analysis as a built in feature. In using these kinds of software, it will be important for you to keep the basic mathematical picture and physical problem formulation in mind. In order to have a well defined problem you will need to specify the following,

- The geometry of the physical domain. In this chapter, our geometry was usually simple (e.g. a line) but in software you can easily have any shape. The software will break your geometry up into a mesh of a number of small elements. For heat conduction, the default meshing process will often be sufficient. In complex problems, meshing can become an art form.
- The equation. While we have discussed the heat conduction equation alone, the software might have additional terms which you can activate, such as internal heat generation.
- Material properties such as α, k, and C. Often there will be a library of built in material properties. You can also set the properties to be temperature dependent if that is important. Alternately, you may want to solve a dimensionless problem and you can set your properties to 1.
- The initial temperature distribution (unless you are only interested in steady state).
- Appropriate boundary conditions. All endpoints, edges, or surfaces of your geometry (whether in 1D, 2D, or 3D respectively) will need boundary conditions specified. The most common conditions would be insulated (no normal temperature gradient), fixed temperature, convection, or fixed heat flow.

While such software solutions are easy to generate, please don't forget about the analysis we have done in this section. You just can't substitute computer simulations with simple analysis. For example, a simple analysis with the resistor analogy can be powerful as a design tool. With resistor analysis you can often see the thermal performance tradeoffs between different parameters. Simple observations can be obscured by

simulation. Design work will have other material and cost tradeoffs involved so having a simple thermal model can be very powerful. When a design has many possible configurations or choices, a simple model that can get you in the right ballpark is more useful than a detailed simulation. Even though the simulations may be easy to run one at a time, they are not always easy to deal with when one has many design choices of different parameters. In some cases, you will make your life more difficult by relying on simulation more than your simple analysis tools. Paper and pencil analysis can also sometimes show you that some effect is small and relatively unimportant and thus can be safely ignored.

In problems where convection is involved as a boundary condition it is important to realize that the value of h is never known precisely. As we will discuss later, it is common for error bars on h to be 25 percent or greater even in well controlled cases. In less controlled cases the error bars can be even greater. While a 3D simulation may be extremely accurate with respect to the conduction model, the final result must be suspect due to uncertainties in h. Thus, a simple model that lumps a complex geometry as a 1D line, for example, might be just as accurate overall as a full simulation.

Finally, it is important to note the ease with which one can make mistakes with software. It is easy to mix up your units, put in the wrong parameters, or make some other seemingly minor software mistake. It is good to have some order of magnitude estimates and qualitative understanding of your problem before resorting to numerical simulation.

7

Diffusive mass transfer

Now we will consider the case of mass transfer, which mathematically is completely analogous to heat transfer. In this chapter we consider an inert and dilute chemical species mixed with a fluid. We may think of this as a little food coloring added to water. The notion of dye implies something passive (no chemical reactions) and we have a way to measure their concentration. We will consider a species which is at low enough concentration that it does not influence the properties of the bulk fluid. Analysis of concentrated fluid mixtures can be quite complex and interesting - but that is a topic for another day.

Through our discussion I will refer to the dye as comprised of molecules; the microscopic physical picture is helpful in our discussion and for physical intuition. Even though we will talk of molecules, it is important not to forget that our analysis is still in the continuum realm. Over any length scale of interest, there are many many many dye molecules in that volume - there are just far fewer dye molecules than water molecules.

Note that while we might be thinking of dye in a fluid, for now we will consider only a stationary fluid (it might as well be a solid). This chapter will set the stage for the next chapter where we will start to see what happens when the fluid starts moving. To start our study, we will only consider mixing by molecular diffusion and later we will consider convective motion of the fluid.

7.1 Diffusive mass transfer

We will take the fluid which our dye is dissolved in as being stationary. The velocity of the fluid is zero everywhere. We define the concentration, c, to be the number of dye molecules per unit volume. If we define an arbitrary volume of interest, then the total number of dye molecules, N, inside that volume is,

$$N = \int c dV. \tag{7.1}$$

Since the dye molecules do not chemically react their number must be conserved. If we consider an arbitrary volume then the time rate of change of the number of molecules stored inside the volume must equal the rate that dye molecules are entering and leaving the volume.

$$\frac{dN}{dt} = \dot{N}_{\text{in}} - \dot{N}_{\text{out}}. \tag{7.2}$$

7.1.1 Mass flow across a surface

Just like in the thermal problem, we must consider the flux of dye to be a vector quantity, $\mathbf{j}$. The units of flux are number of dye molecules per unit time per unit area. The dye molecules can move through the fluid by molecular diffusion while the fluid remains stationary. If we take the surface of an arbitrary volume, and draw a surface normal vector $\mathbf{n}$ to point outward, then $\mathbf{j} \cdot \mathbf{n}$ is the rate that dye molecules cross the surface at that point. Thus, the net rate that the dye molecules flow across the entire boundary of a closed volume is given by integrating the mass flux vector over the whole surface. If we consider our surface, S, which encloses our entire volume of interest the net rate that dye molecules cross that surface is

$$\dot{N}_{\text{in}} - \dot{N}_{\text{out}} = -\int \mathbf{j} \cdot \mathbf{n} dS. \tag{7.3}$$

The minus sign is there because the convention is to define the normal vector to point outwards from a closed surface.

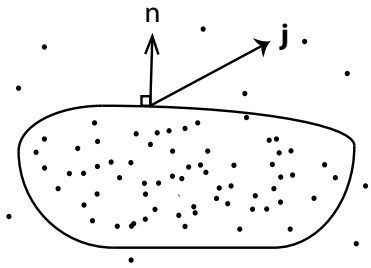


Figure 7.1 Schematic of mass flux and normal vector on an arbitrary, closed surface. The concentration of dye inside the volume is sketched to be higher than outside. This case would have a net flux outward from the volume, decreasing the total number of dye molecules inside.

7.1.2 Conservation of mass: dilute species

If we now consider a closed, arbitrary volume in the material, Equation 7.2 written in vector form is,

$$\frac{d}{dt}\int c dV = -\int \mathbf{j}\cdot\mathbf{n} dS.$$

This equation makes the statement that the time rate of change of the number of molecules inside the volume must equal the net rate that molecules cross the surface. For now, we consider mass transfer in a stationary supporting material and our volume of integration remains fixed and does not move. If the volume is fixed, on the left hand side the order of the derivative and integral can be switched,

$$\int \frac{\partial c}{\partial t} dV = -\int \mathbf{j}\cdot\mathbf{n} dS.$$

Applying the divergence theorem to the right hand side converts the surface integral to a volume integral. Grouping the two volume integral terms together yields,

$$\int (\frac{\partial c}{\partial t} + \nabla\cdot\mathbf{j}) dV = 0.$$

Since the region of integration is arbitrary, the integrand must be zero and our integral equation can be written in differential form as,

$$\frac{\partial c}{\partial t} + \nabla\cdot\mathbf{j} = 0. \tag{7.4}$$

The equation is analogous to the conservation of energy equation derived in the previous chapter. The physical interpretation is also the same as in conduction heat transfer; when all the mass flux vectors flow away from a point, the divergence is positive and the local rate of change of concentration with time decreases.

7.1.3 Fick's law

Just like in the thermal problem, the conservation law alone does not close the problem. Conservation provides a single equation that relates the concentration and the three components of the mass flux vector. We still need a way to connect the flux vector to the concentration field.

Fick's Law is the analogous constitutive law to replace Fourier's law. Fick's Law states that

$$\mathbf{j} = -\mathcal{D}\nabla c. \tag{7.5}$$

The law states there is a linear relationship between diffusive mass flux and concentration gradients. Concentration has units of number of molecules per unit volume. The mass flux has units of number of molecules per unit area per unit time. The diffusivity $\mathcal{D}$ has units of m^2/s and plays a similar role as the thermal diffusivity in conduction. We treat $\mathcal{D}$ as a material property that we can look up. Typically the constant is quite small if the medium is a liquid. For a small molecule in water $\mathcal{D} \sim 10^{-9}$ m^2/s. For larger molecules it can be an order of magnitude smaller. It is important to remember that Fick's law is a constitutive law which is not necessarily valid for non-dilute systems.

7.1.4 Diffusion equation

The use of Fick's law in our conservation equation yields,

$$\frac{\partial c}{\partial t} = \nabla \cdot (\mathcal{D}\nabla c).$$

When the diffusivity is a constant, which is often but not always the case, our equation simplifies to,

$$\frac{\partial c}{\partial t} = \mathcal{D}\nabla^2 c. \tag{7.6}$$

The equation is exactly the same as we derived in chapter on conduction. It is a single equation for the concentration at every point in space and time. It can (in principle) be solved when provided an initial condition and appropriate boundary conditions. If you have a solution to the heat equation, you can use it for this mass diffusion equation. This equation is appropriate only for stationary media.

7.2 Physical picture of mass transfer

We have discussed diffusion as a continuous problem and considered that the concentration is a local continuum quantity. If we get our imaginary microscope we see dye particles that are moving about inside the fluid. Each individual dye particle undergoes a wayward path. The dye particle is constantly colliding with the randomly jiggling and wiggling fluid molecules. All this thermal motion of the fluid causes the dye particle to change direction and speed often. The path of any particle is very random and cannot be predicted. We can however, speak of a statistical average of the random motion.

The random walk is often called the drunken sailor problem. The drunk sailor walks out of the bar has no idea where he is going and does not remember where he has been. Every step is taken at complete random. I do not know why we are so quick to stereotype sailors, but that is beside the point. Some of his steps will be small, some long; some to the left, some to the right. A simulation of this situation is easy to do. Simply start at known x, y coordinate location and with each time step, move a random distance in both x and y. In doing so we use the Gaussian, or normal, distribution to determine the direction and magnitude of each step. Every time we run the simulation we will get something different, a sample is shown in Figure 7.2.

Since each simulation is different we can repeat the experiment many times and ask the question, on average how far away from the bar is the sailor? If we plot the total distance squared from the bar versus time for one situation and the average of many simulations on log-log coordinates we obtain a result as shown in Figure 7.2. We find that in this case that an individual sailor gets further from the bar in a random way, but that on average r^2 increases linearly with time. To go twice the distance takes 4 times as long. This is the same result

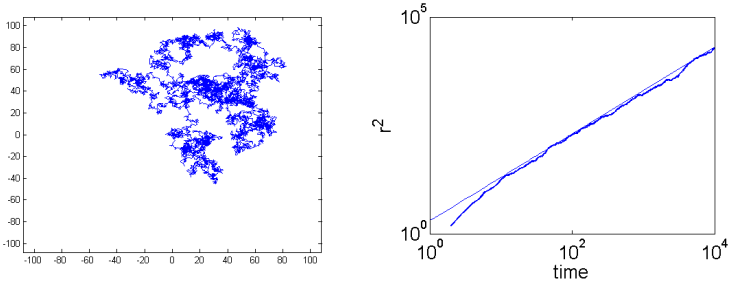


Figure 7.2 On the left, an example of a random path taken by the drunken sailor. The sailor starts at zero,zero and at each instant in time takes a step that has random direction and magnitude. On the right, distance squared, r^2, as a function of time averaged for 100 random walking particles. The straight line is a linear relationship for reference.

we found in the heat conduction problem. In fact, if we conducted a more careful analysis we could show an exact equivalence between the diffusion equation and the random walk in the continuum limit.

The random motion that we have just discussed is also known as Brownian motion. In 1828 Robert Brown noticed that small pollen grains in a liquid had paths as shown in Figure 7.2. Einstein considered this problem in 1905 (the same year that he wrote the theory of relativity) and put the motion of such particles on a theoretical basis. He derived an expression that allowed one of the first accurate measurements of Avogadro's number and other molecular properties.

Why the flux should be proportional to the concentration gradient can be "derived" from this picture of the random walk. Imagine a 2D box and drawing a vertical line along the center, Figure 7.3. It seems reasonable to say that the rate that particles on average cross this imaginary line from left to right would be proportional to the concentration of particles on the left side of the line. Likewise, the rate that particles cross the imaginary line from right to left is proportional to the concentration of particles on the right side of the line. The *net* flux across this imaginary line is the difference of the two values. If the concentration is the same on the left and right side of the line, there is no *net* flux in any direction - there is no concentration gradient. If the concentration is not uniform as in Figure 7.3, then it should be clear that if each

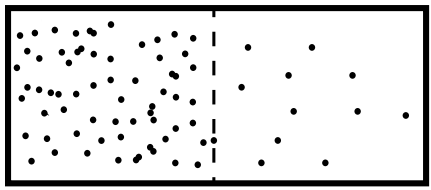


Figure 7.3 Box where the left has higher concentration than the right. .

particle is undergoing a random walk within the box that the left side will be losing particles while the right side will be gaining them. All the particles are acting the same (on average) there are just more particles on the left when we start. Ultimately we would expect the particles to be evenly distributed throughout the box, just based on probability. Given enough time the particles can be at any random location in the box.

If the concentration is not uniform, then the concentration on the right, a distance dx away, is given by a Taylor series as

$$c(x + dx) = c(x) + \frac{\partial c}{\partial x} dx.$$

where $c(x)$ is the concentration on the left and $c(x + dx)$ is the concentration on the right. If the net flux is proportional to the difference in concentration on the left and right side of the line, the net flux would proportional to the concentration gradient.

7.3 Boundary conditions

Just as with heat transfer, we need boundary and initial conditions to solve the mass diffusion equation. My equation is good for mass diffusion in the object of interest. The boundary conditions are the mathematical expression of what is happening just *outside* the object where we are solving the equation.

There are some common classes of boundary conditions which are the same as in heat tansfer. The first is **fixed concentration**. This

condition simply means that we know and prescribe the concentration along the boundary. The second condition is **insulating**. Here, we set the mass flux through the surface to be zero $\mathbf{q} \cdot \mathbf{n} = 0$. This condition would be realized at an impenetrable wall. In terms of the temperature, Ficks's law says the insulating condition is $\nabla c \cdot \mathbf{n} = 0$. We could also **prescribe the mass flux** at the boundary, which would be stated as $-k\nabla c \cdot \mathbf{n} = q$. We can have other variations of boundary conditions that are analogous to those in heat transfer.

7.4 Summary

Heat and mass diffusion are mathematically identical and physically have many similarities. One difference for our intuition is that mass transfer is generally a pretty slow process. If you consider something like sugar mixed in water, the diffusivity is $D = 6 \times 10^{-10}$ m^2/s. In the heat conduction chapter we discussed that you could estimate the time it takes diffusion to occur over some distance as $t \sim L^2/D$. Using this argument and assuming your coffee cup is about $L \sim 10$ cm then it would take about half a year for sugar to mix within your coffee cup. If you wanted the molecules to diffuse over a length scale of your height, you might need to wait 200 years. Diffusion is slow at the human scale. If you consider a single cell, which might have a size of 10 microns (1 micron is 10^{-6} m), then we get diffusion times scales of on the order of a second. Diffusion is not so slow if you happen to be really small. When we observe fluid mixing in our everyday life, it is most certainly dominated by convection. In the next chapter we will begin to describe conservation laws in moving fluids.

8

Convective mass transfer

In this chapter we again consider an inert and dilute chemical species mixed with a fluid. In the previous chapter we considered diffusion only and we saw that diffusion is generally very slow (at least at the human scale). A common analogy for mass diffusion is the example of opening a bottle of perfume at the front of the room. If I did this, the front of the class would smell it first, then it will spread across the room toward the back. The behavior would seem "diffusion-like" but it is not really mass diffusion. The perfume process would be driven by air motion in reality. Even if the air in the room feels still, it is not. Slight currents dominate the slow process of molecular diffusion. In the perfume example we might have to wait years for diffusion to do the trick while convection in seemingly still air takes less than a minute.

We will use the example of a passive dye to introduce how fluid flow can dramatically change the behavior and analysis. Fluid flow also greatly impacts heat transport in liquids and gases. However, there are some details with energy transport that can be a little subtle so we will postpone that discussion and pick up heat movement by convection in a later chapter.

8.1 Convective mass transfer

The formulation derived in the last chapter is useful when the supporting medium is stationary - such as a solid. However, when fluids are involved it is often not the case that things are still. If you drip a little food coloring in water very carefully, you will probably see that after

it sinks to the bottom of the cup it just sits there. You would have to wait quite some time for the dye to fully mix by molecular motion. Give the water a quick flip with a spoon and mixing occurs seemingly in an instant. In general, at the scale of everyday things that you and I are intuitively familiar with, mixing by convection in a fluid is much more efficient than by molecular diffusion. When we observe mixing in our everyday life, it is most certainly dominated by convection. In this section we will start to describe how to describe conservation laws in moving fluids.

In describing fluid flow, we will refer to the carrying fluid's velocity field. We will describe the velocity vector field $\mathbf{v}(t, x, y, z)$ as a function of space and time. By convention, we will refer to the x, y and z components of the velocity vector as $\mathbf{v} = [u, v, w]$. For the purposes of this chapter we will assume by some magic that the fluid's velocity field is known. In the next chapter and for much of the rest of this book we will discuss how to calculate that velocity field - a non-trivial matter.

8.2 Material point of view - be one with the fluid

Imagine a river which meanders through the countryside. The river has some wide regions with very slow flow and some narrow regions with faster flow along with some rapids and waterfalls. The river however, is in a state that is constant with time and does change from hour to hour or day to day. Imagine you are sitting on the bank of this river studying the flow of water. Your reference frame is fixed and everything seems constant and nothing changes if you come back the next day to the same spot. If you measured any property of the river at your particular location, and you would see the same value every day and might be tempted to say nothing is changing. The river is static. Now imagine you are on a raft studying the same river. The river seems anything but steady and constant. You are constantly speeding up, slowing down, twisting left and right, and generally being thrown about. From the raft you would surely report that the river is dynamic. Whether some property of a flow seems constant in time or not, is a matter of whether you like to sit on the river bank or go with the flow.

In formulating our conservation laws, it is useful to take a point of view of the fluid. In analyzing flow we often will refer to a "fluid parti-

cle". This is a nomenclature that is common, but perhaps misleading. There are no particles per se in our description of the flow. By a fluid particle, we mean that we attach an imaginary massless membrane to a fixed amount of fluid. We then watch where this fluid blob goes as it flows and deforms. The fluid particle is considered to be composed of the same material for all time. This consideration is not literal from a molecular point of view, it is surely true that the random motion of molecules means that individual molecules are exchanged from one fluid particle to the next.

8.3 The material derivative

Let us continue with the river example and quantify the idea of change from the fluids perspective. Imagine that upstream, someone is illegally dumping some pollutants and you are tasked with figuring what is going on. Let's imagine a case where the river water flow is constant in time. In principle, if you had enough sensors you could describe the concentration field of the pollutant at every point in space and time; i.e. $c(x, y, z, t)$. This is difficult to do, so you set about to do some more localized sampling.

First, you decide to fix your reference frame by standing in one spot on a river bank. For this point of view we measure the rate of change of pollutant by simply taking the time derivative of the measured concentration, $\partial c/\partial t$. Since the river's flow is constant, if you measure the concentration changing in time we would assume this must have something to do with changes in the rate that pollutants are dumped in the river upstream.

Now imagine that we decide to do some sampling of the river on a small boat. We get on a small boat and drift down the river, just going with the flow. If we monitored the concentration of pollutant from the perspective of the flow, we would sense something different. If we are upstream of the source, we sense no pollutant. As we pass the source, we would presumably measure an increase in concentration. Now that we have become a fluid particle the rate of change of concentration with respect to time *from the perspective of the fluid*, is different than in the fixed frame. We will find it useful to describe the rate of change of something with respect to time from the perspective of the fluid.

Even though we measure a different rate of change depending on the perspective, it is important to remember that there is only one concentration field. If you are measuring from the river bank and me from the boat, we better get the same instantaneous measurement at the time I drift over your fixed sensor. We are making measurements of the exact same thing and ultimately our perspectives must line up.

8.3.1 Derivation in 1D

Let's start with a simplified example. Imagine the pollutants in a particular region of the river follow a linear and steady concentration field that only varies in the downstream coordinate x. If you walk along the river in this location and keep taking measurements of the concentration field from the river bank you find a region that concentration increases linearly, Figure 8.1. For this example, the concentration field at the location of change is

$$c(x) = c_0 + \Delta c \frac{x}{L}$$

where Δc is the increase in concentration over distance L. The slope of the concentration field in this region is constant,

$$\frac{dc}{dx} = \frac{\Delta c}{L}$$

The measurement from the bank shows the concentration field to be fixed in time. If I return to the same point on the river bank tomorrow, I will get the same measurement. However, this is a straight river with constant flow from day to day and at every location. In the boat we just float right down the middle at constant velocity. If we take measurements from the boat moving at constant velocity, u, then from that perspective we would sense an increase in concentration with respect to time. Since we traverse distance L in a time of $\Delta t = L/u$ then from the boat's perspective the time rate of change is,

$$\frac{dc_{\text{boat}}}{dt} = \frac{\Delta c}{\Delta t} = u\frac{\Delta c}{L},$$

which can be written as,

$$\frac{dc_{\text{boat}}}{dt} = u\frac{dc}{dx}.$$

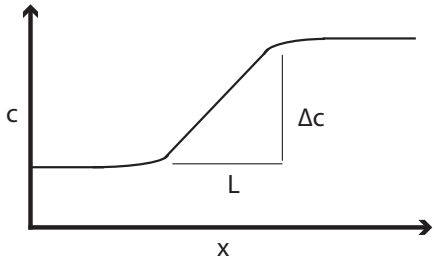


Figure 8.1 Example concentration field, $c(x)$, with a region of linear change in concentration. The field is considered steady in time. If you transport yourself through this *fixed* concentration field from left to right you will sense an increase in concentration with respect to time. The rate of change you sense will depend upon the speed with which you move through this field.

The above expression should make sense, the faster we move through the fixed concentration field the higher rate of change we will see from the boat's perspective.

We can also get the same result appealing to the chain rule. The boat follows a simple path,

$$x_{\text{boat}}(t) = x_o + ut$$

The stationary concentration field, $c(x)$ would be seen from the boat as a function of time by $c(x_o + ut)$. Taking the derivative with respect to time using the chain rule yields,

$$\frac{dc(x(t))}{dt} = \frac{dc}{dx}\frac{dx}{dt} = u\frac{dc}{dx}$$

8.3.2 General derivation

To more formally describe change from the perspective of the fluid, we take the variable $\mathbf{a}$, to be the location of a fluid particle at time $t = 0$

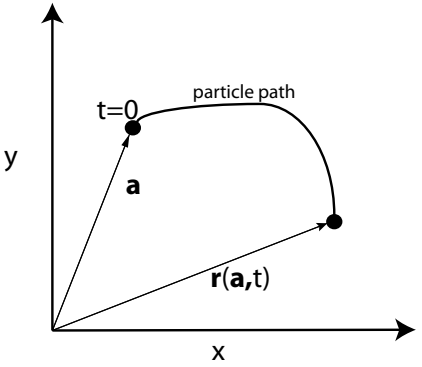


Figure 8.2 Schematic of the material point of view. The vector **a** describes where the chosen particle is at $t = 0$. The vector **r** describes where the particle is a little later.

and $\mathbf{r}(\mathbf{a}, t)$ is the location of the same particle at some later time t; see Figure 8.2. If we knew the function $\mathbf{r}(\mathbf{a}, t)$ we would know everything about the flow. Think about this for a minute, the notation above is perhaps not intuitive at first glance. We are defining everything about the flow based on tracking where each and every blob of fluid goes.

In this frame, the velocity of a fluid particle is defined as

$$\left.\frac{\partial \mathbf{r}(\mathbf{a}, t)}{\partial t}\right|_{\mathbf{a}=\text{const}} = \mathbf{v}$$

The notation above says we take the derivative of **r** and hold **a** constant. It is perfectly valid for us to define the velocity as a function of **a**, "material" or as a function of **r** "space". Note that this expression is a vector equation as denoted by the boldface notation.

To demonstrate the material derivative, consider the concentration field, c that is a function of space and time. Normally, we would think to write the concentration in our fixed frame as $c(x, y, z, t)$. However, we could equivalently write the concentration field through the mapping,

$$c(\mathbf{r}(x, y, z, t)) = c(x(\mathbf{a}, t), y(\mathbf{a}, t), z(\mathbf{a}, t), t)$$

where $x(\mathbf{a}, t)$ is the first component of the vector **r**. While this mapping seems confusing written as an equation, intuitively the idea is simple. If I am on the bank of the river sampling the concentration at a particular

point, and you pass through that point on your raft, at the instant you pass through my point, our measurements must agree.

The derivative with respect to time for a fixed fluid particle is found using the chain rule,

$$\left.\frac{\partial c(\mathbf{r}(\mathbf{a},t),t)}{\partial t}\right|_{\mathbf{a}=\text{const}} = \frac{\partial c}{\partial t} + \frac{\partial c}{\partial \mathbf{r}}\frac{\partial \mathbf{r}}{\partial t}.$$

Note that $\mathbf{v} = \frac{\partial \mathbf{r}}{\partial t}$ is the same as the fluid particle velocity. In vector form, $\frac{\partial c}{\partial \mathbf{r}} = \nabla c$, the gradient of the concentration field. Thus,

$$\left.\frac{\partial c(\mathbf{r}(\mathbf{a},t),t)}{\partial t}\right|_{\mathbf{a}=\text{const}} = \frac{\partial c}{\partial t} + \mathbf{v}\cdot\nabla c = \left(\frac{\partial}{\partial t} + \mathbf{v}\cdot\nabla\right)c = \frac{Dc}{Dt} \quad (8.1)$$

The notation $D/Dt = \partial/\partial t + \mathbf{v}\cdot\nabla$ is commonly used for compactness and is called the material derivative. The material derivative's purpose in life is to tell us the rate of change of something in time from the perspective of the fluid. It captures the two effects; the field itself changing with time and a fluid particle moving about in space where the field changes from point to point. If we expand into component form we have

$$\frac{Dc}{Dt} = \frac{\partial c}{\partial t} + \mathbf{v}\cdot\nabla c = \frac{\partial c}{\partial t} + u\frac{\partial c}{\partial x} + v\frac{\partial c}{\partial y} + w\frac{\partial c}{\partial z}.$$

If you got a little lost in the detail of the 3D derivation, don't be dismayed. The 1D example captures all the essential ideas, we are just generalizing our derivations.

8.4 Liebniz and Reynold's transport theorem

We can also discuss the material point of view in integral form. To do so we need the Leibniz rule from calculus. The Leibniz rule allows us to calculate the derivative of an integral when the limits are changing with time. In 1D this rule is,

$$\frac{d}{dt}\int_{a(t)}^{b(t)} f(t,x)dx = \int_a^b \frac{\partial f}{\partial t}dx + \frac{db}{dt}f(t,x=b) - \frac{da}{dt}f(t,x=a).$$

The three terms represent the change in the integral in time due to the change in the function itself, the change in the upper limit of integration and the change in the lower limit of integration. Here f is just some arbitrary function - this is a mathematical statement.

In three spatial dimensions this theorem changes to;

$$\frac{d}{dt}\int_{V(t)} f(\mathbf{x},t)dV = \int_{V(t)} \frac{\partial f}{\partial t}dV + \int_{S(t)} f\mathbf{v}\cdot\mathbf{n}dS. \qquad (8.2)$$

In this expression instead of using d/dt to denote the rate of change of the location of the interface, we have used the velocity, $\mathbf{v}$, of the surface of integration. We use the subscript $V(t)$ on the integral to remind us that the volume of integration can be changing with time. The above theorem says that the rate of change of a scalar field inside a volume, V, is composed of two parts; the change due to the change of f inside the control volume and the change due to the movement of the surface. When Equation 8.2 is applied to a material fluid particle, then the velocity on the surface of integration is equal to the fluid velocity.

Also note that use of the divergence theorem would allow us to write the second term as a volume integral, namely,

$$\frac{d}{dt}\int_{V(t)} f(\mathbf{x},t)dV = \int_{V(t)} \left(\frac{\partial f}{\partial t} + \nabla\cdot(f\mathbf{v})\right) dV. \qquad (8.3)$$

When Equation 8.2 or 8.3 is applied in fluid mechanics it is often called Reynold's Transport Theorem. While it can also be derived a few different ways, we will treat it as simply a mathematical theorem from calculus. The usefulness of this theorem will be seen in the next section where we consider conservation laws. The connection to the material derivative D/Dt will become apparent in the next chapter.

8.5 Conservation of mass: the fluid

Now lets consider conservation of mass; we consider the fluid itself and the dye separately. Remember we will be interested in the limit where the dye is so dilute it has no impact on the fluid it is dissolved in. In this limit we just ignore the dye for deriving conservation of mass for the fluid. The total fluid mass inside a material volume, V(t), is given as

$$M = \int_{V(t)} \rho dV,$$

where $\rho(\mathbf{x},t)$ is the density field. The subscript $V(t)$ reminds us that the volume of integration is not fixed in time and may be changing shape as

the flow deforms the material. Our volume $V(t)$ is going with the flow and following the material for all times. Since our volume of integration was taken to be a material volume (i.e. it encloses the same amount of material at all times) and since mass is conserved then $\frac{dM}{dt} = 0$. Using the Transport Theorem (Equation 8.3),

$$\frac{dM}{dt} = \frac{d}{dt}\int_{V(t)} \rho dV = \int_{V(t)} \left(\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) \right) dV = 0$$

Since the volume in our analysis was arbitrary and since the integral is always equal to zero then the the integrand must be zero,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0. \tag{8.4}$$

This equation must be true point-wise in the fluid and is the differential statement of mass conservation. By rearrangement,

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{v} = 0. \tag{8.5}$$

This equation says that the density of a fluid particle can only change if the velocity field has non-zero divergence. Divergence of the velocity field is a measure of the net rate of fluid flowing into/out of a point. If the velocity vectors are all pointing to one location, then the flow must be accumulating mass at that point and the density should be increasing. If the velocity vectors all point away from a point in the flow, the density should be decreasing.

For an incompressible flow with constant density we have the simplified relation,

$$\nabla \cdot \mathbf{v} = 0.$$

This assumption of incompressibility is one that we will use throughout the course. For a liquid the assumption of incompressibility is a good one. I challenge you to compress some water to any significant degree. For a gas the assumption is questionable. It turns out that a rule of thumb people use is that if the flow velocity is less than about 1/3 of the speed of sound in a gas, then the assumption of incompressible flow is basically fine. For air this means we can safely treat cases with flow velocities on the order of 100 m/s or less as an incompressible flow. Flow around a car is incompressible, but flow around an airplane is not. Compressible flows have many interesting features that will be beyond the scope of our first course.

It is also important that we refer to a flow as incompressible and not the fluid. For the flow of air, we have many examples of incompressible flow despite the fact that air is quite compressible.

8.5.1 Conservation of mass: dilute species

Now let's consider conservation of the dilute passive dye. We take a closed, arbitrary volume *moving with the fluid* and consider the conservation of a trace dye species within that moving volume. The conservation law states that,

$$\frac{d}{dt}\int_{V(t)} c dV = -\int_{S(t)} \mathbf{j}\cdot\mathbf{n} dS,$$

The subscripts $V(t)$ and $S(t)$ remind us that the volume of integration is not fixed in time and is changing shape as the flow deforms the carrier fluid. Using the Transport Theorem (Equation 8.3) for the left side of the equation and the divergence theorem for the right side we obtain,

$$\int_{V(t)} \left(\frac{\partial c}{\partial t} + \nabla\cdot(c\mathbf{v})\right) dV = -\int_{V(t)} \nabla\cdot\mathbf{j} dV$$

We can group all the integrands together and since the volume in our analysis was arbitrary the integrand must always be zero,

$$\frac{\partial c}{\partial t} + \nabla\cdot(c\mathbf{v}) = -\nabla\cdot\mathbf{j}.$$

Substituting in Fick's law from the previous chapter and assuming a constant diffusivity,

$$\frac{\partial c}{\partial t} + \nabla\cdot(c\mathbf{v}) = \mathcal{D}\nabla^2 c. \tag{8.6}$$

Now, a new term has appeared in the equation $\nabla\cdot(c\mathbf{v})$ which represents convection of the dye molecules by fluid motion.

Let's think about this equation for a minute. In the case where there is no fluid motion, $\mathbf{v} = 0$, we are back to the classic diffusion equation. In another limit, let's assume there is no molecular diffusion. As we discussed earlier in the chapter, molecular diffusion is very slow and so in many cases ignoring it is not an unreasonable assumption. In that case the equation would be,

$$\frac{\partial c}{\partial t} = -\nabla\cdot(c\mathbf{v}).$$

The product $c\mathbf{v}$ is a vector which gives the convective flux or rate that dye is carried by the fluid per unit time, per unit area at a point. If you recall that c has units of number of molecules per unit volume, you can check the units of $c\mathbf{v}$ yourself. Now recall that the divergence operator gives us a measure of whether a vector field is converging or diverging at a point. If the convective flux vectors are converging at a point, then the concentration field must be locally increasing in time. If the convective flux is flowing outward from a point in space, the local concentration should be decreasing.

By rearrangement, Equation 8.6 could be written as,

$$\frac{\partial c}{\partial t} + \mathbf{v} \cdot \nabla c + c \nabla \cdot \mathbf{v} = \mathcal{D} \nabla^2 c.$$

Substituting in the conservation of mass from the previous section we can replace the divergence of the velocity field as,

$$\frac{\partial c}{\partial t} + \mathbf{v} \cdot \nabla c - \frac{c}{\rho} \frac{D\rho}{Dt} = \mathcal{D} \nabla^2 c,$$

which can be rewritten using the material derivative as

$$\frac{Dc}{Dt} - \frac{c}{\rho} \frac{D\rho}{Dt} = \mathcal{D} \nabla^2 c. \tag{8.7}$$

This last equation has an interesting interpretation. The material derivative D/Dt describes the rate of change of concentration from the perspective of the fluid. The operator is accounting for convection by fluid motion. The right hand side is the diffusion term. The term on the left $\frac{c}{\rho} \frac{D\rho}{Dt}$ describes the concentration of dye (number per unit volume of fluid) changing because the carrier fluid itself is being compressed or expanded. If I have a container containing a fixed number of dye molecules then I compress the closed contained to decrease the total volume the dye concentration has increased even though the number of molecules has not.

For incompressible flow (liquids are really pretty incompressible) the density doesn't change and thus the mass transfer equation is simplified to

$$\frac{Dc}{Dt} = \mathcal{D} \nabla^2 c, \tag{8.8}$$

where the equation is "exactly" what we found for diffusive transport with no fluid motion only we must remember that we have replace the

material derivative with the normal partial derivative with time on the left side of the equation.

It is also interesting to note that in this form, if there were no molecular diffusion then the equation would reduce to

$$\frac{Dc}{Dt} = 0$$

which means that the fluid particle cannot change its concentration, the particle keeps it's initial value and the dye just moves it with the fluid. The equation $Dc/Dt = 0$ for a non-diffusive dye is at it's essence the very definition of the material derivative - something that is just transported by the flow has zero material derivative. It is this physical interpretation of this mathematical operator that I want to stress. The material derivative is simply the change with respect to time from the perspective of the moving fluid.

8.6 Dimensionless formulation

Let's consider a specific problem. Consider a solid cylinder of radius R made of solid "dye" in a fluid flow. The incompressible flow of the fluid is steady in time. Far from the object the flow has velocity U and near the object the flow field would be more complex as the fluid skirts around the cylinder. Assume, though, by some sorcery we have calculated the complete velocity field, $\mathbf{v}$. Upstream, the fluid approaching the cylinder has a zero dye concentration. The solid dye as it dissolves sets the concentration at the fluid/solid interface to be c_0. To solve the problem for computing the dye concentration, we would state the formulation as

$$\frac{\partial c}{\partial t} + \mathbf{v} \cdot \nabla c = \mathcal{D}\nabla^2 c,$$

where

$$c(r = R) = C_0, \qquad c(r = \infty) = 0.$$

The initial concentration everywhere at time zero is $c = 0$. We define the dimensionless coordinates to be $\tilde{x} = x/R$ and $\tilde{y} = y/R$. The dimensionless concentration is $\tilde{c}/C_0$ and time is made dimensionless by a (for

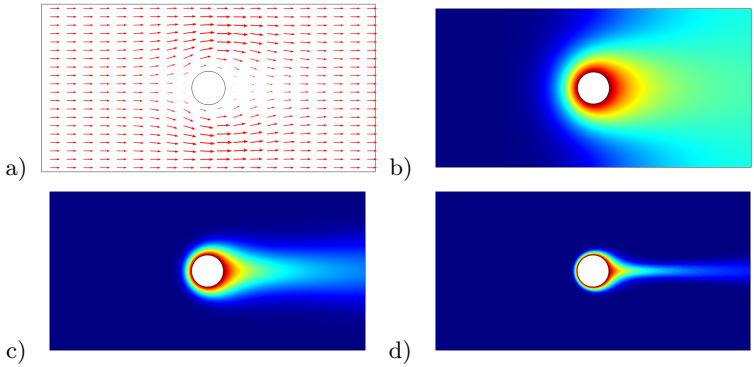


Figure 8.3 a) Velocity field for flow around a cylinder. Concentration field where fluid enters with concentration of zero on the left and the cylinder surface sets a concentration in the fluid. b) Pe = 1, c) Pe = 10, d) Pe = 100.

now) arbitrary constant $\tilde{t} = t/t_0$. Making these substitutions yields,

$$\frac{C_0}{t_0}\frac{\partial \tilde{c}}{\partial \tilde{t}} + \frac{C_0 U}{R}\tilde{\mathbf{v}} \cdot \nabla \tilde{c} = \frac{\mathcal{D} C_0}{R^2}\nabla^2 \tilde{c}.$$

setting $t_0 = R/U$ we get,

$$\frac{\partial \tilde{c}}{\partial \tilde{t}} + \tilde{\mathbf{v}} \cdot \nabla \tilde{c} = \frac{\mathcal{D}}{UR}\nabla^2 \tilde{c}. \tag{8.9}$$

The chosen time scale was the time it takes to convect something the distance of the cylinder radius. This time scale was a convenient choice as it made the coefficient in the equation go to 1.

The parameter $UR/\mathcal{D}$ is called the Peclet number (pronounced, peck-lay) and is a dimensionless measure of the strength of convection to the strength of diffusion. A high Peclet number occurs when the flow is fast around a large object. A high Peclet number means that the fluid carries the dye downstream. We might expect at high Peclet number the dye is mostly concentrated in a wake downstream of the cylinder. For a small Peclet number diffusion dominates, and in this limit we would expect the dye to work its way upstream against the incoming flow. Physically, since $\mathcal{D} \sim 10^{-9}\mathrm{m}^2/\mathrm{s}$ for small molecules in water, a 1 mm/s flow around a 1 mm object would have a Peclet number of about 1000. Thus, in water it is only flows truly at the microscale that have low Peclet numbers.

9

Conservation of momentum in a fluid

The basic conservation law for momentum is developed in this chapter. The law takes a form different than the law for particles that you may be used to, but in the end the equations are nothing more than Netwonian mechanics. The analysis in this chapter does not actually assume we have a fluid, the laws work equally well for solids or materials that are between fluid and solid. However, the fact that the fluid moves, flows, and deforms makes the development of our laws seem different than you might see in a solid mechanics course. We will want to use our conservation laws to set up equations that (at least in principle) can be used to calculate the velocity vector $\mathbf{v}(t, x, y, z)$ as a function of space and time. It should be stressed that the velocity is not the velocity of individual molecules, but is the averaged velocity for all the molecules at a point. The velocity is that of the fluid in a continuum sense.

In the last chapter we discussed how fluid motion can carry dye molecules around in a fluid flow. We applied conservation of mass and our vector calculus theorems to work out how an equation for mass being transported by a fluid flow. Now we will apply the same strategy to conservation of momentum. One difference we find with momentum transport is that the fluid motion that transports momentum around also has momentum itself (think about that for a minute). This realization introduces a non-linearity to the momentum equation that is the source of much of the interesting behavior we find in fluid flows and the source of the real mathematical difficulty with the equations.

Before we proceed, let's remind ourselves of two important results from the last chapter. We discussed the Transport theorem, a result

that is central to the development in this chapter,

$$\frac{d}{dt}\int_{V(t)} f(\mathbf{x},t)dV = \int_{V(t)} \left(\frac{\partial f}{\partial t} + \nabla\cdot(f\mathbf{v})\right) dV. \tag{9.1}$$

It is important to keep in mind that this theorem is a mathematical result from calculus. We also discussed conservation of mass for a fluid which we derived as,

$$\frac{D\rho}{Dt} + \rho\nabla\cdot\mathbf{v} = 0, \tag{9.2}$$

which for incompressible flows became $\nabla\cdot\mathbf{v} = 0$.

9.1 Transport theorem revisited

Previously we stated the Reynolds Transport Theorem for an arbitrary function f in Equation 9.1. Lets apply this theorem to some quantity which is measured per unit mass, b. We would be interested in knowing about the behavior of the integral of ρb over a volume. Apply the transport theorem and simply substituting $f = \rho b$, we obtain,

$$\frac{d}{dt}\int_{V(t)} \rho b dV = \int_{V(t)} \left(\frac{\partial \rho b}{\partial t} + \nabla\cdot(\rho b\mathbf{v})\right) dV.$$

We can expand the derivatives on the right side to write this expression as

$$\frac{d}{dt}\int_{V(t)} \rho b dV = \int_{V(t)} \left[b\left(\frac{\partial \rho}{\partial t} + \nabla\cdot(\rho\mathbf{v})\right) + \rho\left(\frac{\partial b}{\partial t} + \mathbf{v}\cdot\nabla b\right)\right] dV.$$

This form is convenient since the first term in parenthesis is zero from mass conservation, Equation 9.2. Our final result is then,

$$\frac{d}{dt}\int_{V(t)} \rho b dV = \int_{V(t)} \rho\left(\frac{\partial b}{\partial t} + \mathbf{v}\cdot\nabla b\right) dV = \int_{V(t)} \rho\frac{Db}{Dt} dV, \tag{9.3}$$

which is useful for subsequent analysis. Note that we made no assumptions about incompressibility here, so this relationship is one that is always true. This relationship is convenient because it already separates out conservation of mass.

9.2 Linear momentum

The total linear momentum, $\mathbf{p}$, inside a material volume, V(t), is given as

$$\mathbf{p} = \int_{V(t)} \rho \mathbf{v} dV,$$

where $\rho(\mathbf{x}, t)$ is the density field and $\mathbf{v}(\mathbf{x}, t)$ is the velocity vector field. Note that this is a vector expression as the momentum has an x, y, and z component. In Newton's law we know that we need to be able to take the time derivative of momentum. To take the time derivative wef can use Equation 9.3, simply replacing b with $\mathbf{v}$,

$$\frac{d\mathbf{p}}{dt} = \frac{d}{dt} \int_{V(t)} \rho \mathbf{v} dV = \int_{V(t)} \rho \frac{D\mathbf{v}}{Dt} dV \tag{9.4}$$

This form might make you uncomfortable since this is a vector equation and we only discussed the transport theorem with scalar functions. If it does you could proceed with only the x component of the momentum instead,

$$\frac{dp_x}{dt} = \frac{d}{dt} \int_{V(t)} \rho u dV = \int_{V(t)} \rho \frac{Du}{Dt} dV.$$

We could repeat the same procedure for p_y and p_z. In working through the three components, we would return to the compact vector form of the equation in Equation 9.4.

Note that in our vector notation,

$$\frac{D\mathbf{v}}{Dt} = \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v}.$$

The term $\mathbf{v} \cdot \nabla \mathbf{v}$ takes a little explanation. It is probably easiest to remember what it means if you write the term as $(\mathbf{v} \cdot \nabla)\mathbf{v}$. Expand the part in parenthesis first, $(\mathbf{v} \cdot \nabla) = u\frac{\partial}{\partial x} + v\frac{\partial}{\partial y} + w\frac{\partial}{\partial z}$; which is just the dot product of the velocity vector and the gradient operator. Now apply this operator to the velocity vector to get the components of $(\mathbf{v} \cdot \nabla)\mathbf{v}$. For example, the x component of $(\mathbf{v} \cdot \nabla)\mathbf{v}$ is $(u\frac{\partial}{\partial x} + v\frac{\partial}{\partial y} + w\frac{\partial}{\partial z})u = (u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z})$. The entire operator expanded into component form

is a vector,

$$\mathbf{v}\cdot\nabla\mathbf{v} = \begin{bmatrix} u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \\ u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} \\ u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} \end{bmatrix},$$

where the first, second, and third rows are the x, y, and z components of the vector.

The expression 9.4 allows us to compute the rate of change of momentum of a material volume. Newton's laws requires that this vector must equal the sum of the forces acting on the control volume. Determining the forces acting on the fluid is not a simple matter and will take some work. Before we turn to computing forces, let's make sure we really understand the idea of acceleration in fluid flows.

9.2.1 Interpretation of acceleration

The material derivative of the velocity,

$$\frac{D\mathbf{v}}{Dt} = \frac{\partial\mathbf{v}}{\partial t} + \mathbf{v}\cdot\nabla\mathbf{v},$$

is the expression for acceleration of a fluid particle. The expression says there are two components to the acceleration in a fluid. The first term $\frac{\partial\mathbf{v}}{\partial t}$ has to do with the velocity field changing with time. The second term $\mathbf{v}\cdot\nabla\mathbf{v}$ is acceleration from the perspective of the flow.

Imagine our river again. I go to the river every day to the same point on the banks and measure the river's flow velocity. Day to day, nothing changes and I conclude that at this location that $\frac{\partial\mathbf{v}}{\partial t} = 0$. There is no local acceleration in the flow due to the velocity field changing with time. However, that does not mean the acceleration that the fluid feels is zero. Downstream from our measurement point there is a set of rapids. If I go down the river on the raft, from the perspective of the fluid I certainly feel acceleration as I speed up and slow down through the rapids. This acceleration arises from the $\mathbf{v}\cdot\nabla\mathbf{v}$ term. Now imagine that one day there is a big storm and when I go out to measure the river, the velocity at my special point has increased. Thus it must be true that $\frac{\partial\mathbf{v}}{\partial t} > 0$, at least for some of the time between now and the previous day. So there are two different types of acceleration, one is due

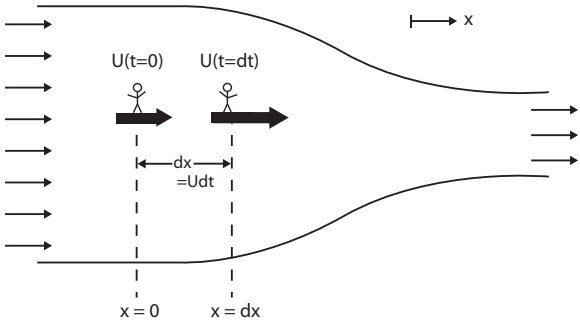


Figure 9.1 Schematic of acceleration in a fluid flow from the perspective of the fluid. A person riding a raft through a narrowing river feels acceleration even if the fluid flow is constant when observed from the banks of the river.

to the velocity field changing in time and the other is from the velocity field changing in space.

We can see that the spatial acceleration must follow a form like $\mathbf{v}\cdot\nabla\mathbf{v}$ in a simple 1D example. Imagine a smooth river with a constant flow in time. The river has a constant depth but there is a region where the banks narrow down in width. Since the cross sectional area of the river has been reduced, the flow velocity must increase in the narrow region, shown in Figure 9.1.

Now let's measure the acceleration of the fluid from the perspective of the flow and the fixed river banks. Let's assume that I am on a raft and you are on the river bank. We are going to measure the acceleration that a raft feels at the same location from our two different perspectives. Even though we are observing from different points of view, in the end our measurements should be the same. If I go right down the center of the river on a raft, I sense the x component of the river flow velocity. If I want to know my acceleration, I simply measure the raft velocity, U, at two instances, $t = 0$ and a short time later, $t = dt$;

$$\text{Acceleration} = \frac{U_{\text{raft}}(t = dt) - U_{\text{raft}}(t = 0)}{dt}$$

Now you are observing from the river bank and want to know the fluid acceleration at the same point. If you watch me go past you could measure my acceleration the same way. However, assume there is no

tracer for you to observe in the flow and you are stuck taking point measurements of the river's flow velocity. The previous formula for acceleration is still valid, however from the bank you can only measure the river velocity as a function of position. The equivalent measurement of the fluid acceleration would would be,

$$\text{Acceleration} = \frac{u(x = dx) - u(x = 0)}{dt}.$$

The distance dx that I would have moved on the raft over time dt is related to the flow velocity as $dx = u\ dt$. Therefore substituting for dt,

$$\text{Acceleration} = u\frac{u(x = dx) - u(x = 0)}{dx}.$$

Now the fraction in this equation is the definition of the spatial derivative of the flow velocity in the limit that dx goes to zero,

$$\text{Acceleration} = u\frac{\partial u}{\partial x}.$$

The expression above is simply the 1D version of $\mathbf{v}\cdot\nabla\mathbf{v}$, the acceleration of a fluid particle due to changes in the velocity field in space.

9.3 $\mathbf{F} = m\ \mathbf{a}$

In particle mechanics, we usually think of Newton's Law expressed as the equation $\mathbf{F} = m\ \mathbf{a}$. However, this is for a constant mass particle and a more general statement would be

$$\frac{d\mathbf{p}}{dt} = \mathbf{F},$$

where $\mathbf{p}$ is the linear momentum and $\mathbf{F}$ is the sum of the forces acting on the particle. So we have developed a way to describe the rate of change of the momentum for a fluid particle. Now we need to work on the forces. We will account for body forces acting on the whole of the material volume, $\mathbf{f}_b$ and surface forces $\mathbf{f}_s$ acting on the boundary separately. Generally then, conservation of momentum can be stated as,

$$\int_{V(t)} \rho\frac{D\mathbf{v}}{Dt}dV = \mathbf{f}_b + \mathbf{f}_s. \tag{9.5}$$

9.3.1 Body forces

Obtaining an expression for the body forces is relatively straightforward. The only body force we will consider in this course is that due to gravity. The gravitational body force is simply the continuous version of $\mathbf{f} = m\mathbf{g}$, namely,

$$\mathbf{f}_b = \int_{V(t)} \rho \mathbf{g} dV \tag{9.6}$$

We could also consider electrostatic forces (if the fluid were charged and subjected to an electric field) or Lorenz forces (if the fluid were passing a current and subject to magnetic fields). These are interesting topics for another day, however their inclusion into our theory would not be difficult. If we know how to calculate the force per unit volume, we only need to integrate this force density over our volume.

9.3.2 Surface forces

Describing the stress along the surface of an arbitrary material volume *inside* the fluid takes a little work. First lets define a stress vector, $\mathbf{s}$, as

$$\mathbf{s}(\mathbf{x}, t).$$

The vector is interpreted as force per unit area at some location on our surface of our arbitrary material volume. The total surface force that is exerted on the enclosed volume is then given as,

$$\mathbf{f}_s = \int \mathbf{s} dS$$

Figure 9.2 shows a schematic where we need to add up, or integrate, all the little surface forces acting over the entire material volume's surface.

The general description of the state of stress within the fluid requires us to introduce the idea of a tensor. A tensor has a structure like a matrix. In three dimensions, a tensor is a 3x3 matrix with 9 components. A tensor is needs to describe the state of stress in a material as there are 9 things that are important. The stress at a location has magnitude and direction; it is a vector with 3 components. However, what happens to the material under this force depends upon which face the stress acts. A stress with only an x component which acts on the surface with the

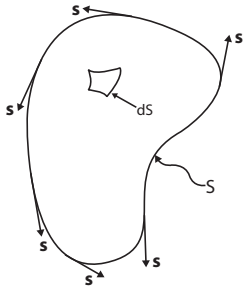


Figure 9.2 The local stress vector, **s**, acts everywhere on the surface, S, of an arbitrary region. Integrating that stress over the surface area gives the net force acting on the surface of the volume.

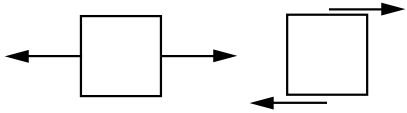


Figure 9.3 Schematic of a square differential element of fluid. A force in the x direction deforms the material differently depending on which face the force acts upon. The element is stretched or sheared, depending upon which face the force acts.

normal vector in the x direction will stretch a chunk of fluid. A stress in the same direction but acting on a face with a normal vector in the y direction will shear a chunk of fluid. This effect is shown schematically in Figure 9.3. Thus the deformation of the material depends upon the direction of the force and which face it acts upon. Thus the state of stress requires more information than can be contained in a vector, so we go to a tensor. We will follow the convention that tensors are boldface capital letters and vectors are boldface lowercase letters. Note that we could spend a whole course on tensor calculus, thus we are covering only the highest of high points here.

We can define a stress tensor with nine components that define the state of stress on an arbitrary element within the fluid. The convention is

$$T_{xz}$$

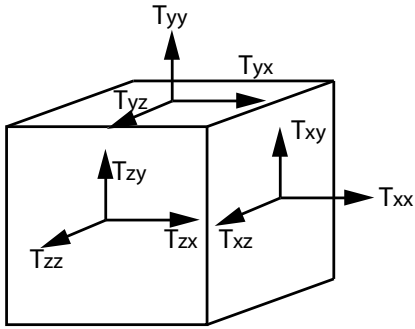


Figure 9.4 Schematic the 9 components of the stress tensor, $\mathbf{T}$.

where x is the direction of the normal plane and z is the direction of the force associated with the stress. The tensor can be represented as,

$$\text{state of stress at point} = \mathbf{T} = \begin{bmatrix} T_{xx} & T_{xy} & T_{xz} \\ T_{yx} & T_{yy} & T_{yz} \\ T_{zx} & T_{zy} & T_{zz} \end{bmatrix}$$

with the components shown schematically in Figure 9.4. Each row of the tensor is for a different face of the cube and each column is for the three components of the force.

Just as we have seen scalar fields and vector fields, the stress tensor, $\mathbf{T}$, is a tensor field, i.e.

$$\mathbf{T}(x, y, z, t).$$

The relationship between the stress tensor at a point, $\mathbf{T}$, and stress vector $\mathbf{s}$ is,

$$\mathbf{s} = \mathbf{n} \cdot \mathbf{T}$$

The dot product projects the tensor onto the surface with a normal vector $\mathbf{n}$ to get the stress vector at that point on the surface. The dot product of two vectors gives a scalar, while the dot product of vector and a tensor gives a vector. To remember what $\mathbf{n} \cdot \mathbf{T}$ means, it is like

the vector-matrix operation,

$$\mathbf{s} = \begin{bmatrix} n_x & n_y & n_z \end{bmatrix} \begin{bmatrix} T_{xx} & T_{xy} & T_{xz} \\ T_{yx} & T_{yy} & T_{yz} \\ T_{zx} & T_{zy} & T_{zz} \end{bmatrix}$$

You can test this yourself by taking a normal vector pointing in the x-direction, n = [1 0 0]. Taking $\mathbf{n} \cdot \mathbf{T}$ with this normal vector returns the first row of $\mathbf{T}$ which are the three forces acting on that face. Fortunately, conservation of angular momentum will require the stress tensor to be symmetric, thus if you forget the order of multiplying the tensor and vector, you will get the right answer anyway. I still to this day confuse (without looking it up) the what components go in the rows and which go in the columns of $\mathbf{T}$ but since the tensor is symmetric I am saved. Also, you might be tempted to confuse $\mathbf{n} \cdot \mathbf{T}$ as being $\mathbf{Tn}$ (i.e. the usual matrix vector multiply), and again symmetry of the tensor will give you the right answer. Since we won't use the tensor notation much outside of deriving equations and since this is likely the first class where you have seen tensors, I'll cut you some slack on keeping everything straight!

The total force exerted on an arbitrary volume element by the surface forces, $\mathbf{f}_s$, is given as simply the integral of the stress vector around the closed surface, i.e.

$$\mathbf{f}_s = \int \mathbf{s} dS = \int \mathbf{n} \cdot \mathbf{T} dS.$$

By the divergence theorem, which also works for tensors just as it does for matrices,

$$\mathbf{f}_s = \int \mathbf{n} \cdot \mathbf{T} dS = \int \nabla \cdot \mathbf{T} dV. \tag{9.7}$$

In component form, the divergence of the stress tensor is,

$$\nabla \cdot \mathbf{T} = \begin{bmatrix} \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \end{bmatrix} \begin{bmatrix} T_{xx} & T_{xy} & T_{xz} \\ T_{yx} & T_{yy} & T_{yz} \\ T_{zx} & T_{zy} & T_{zz} \end{bmatrix} =$$

$$\begin{bmatrix} \frac{\partial T_{xx}}{\partial x} + \frac{\partial T_{yx}}{\partial y} + \frac{\partial T_{zx}}{\partial z}, & \frac{\partial T_{xy}}{\partial x} + \frac{\partial T_{yy}}{\partial y} + \frac{\partial T_{zy}}{\partial z}, & \frac{\partial T_{xz}}{\partial x} + \frac{\partial T_{yz}}{\partial y} + \frac{\partial T_{zz}}{\partial z} \end{bmatrix}$$

Note that the divergence of the tensor returns a vector. So far we haven't done anything, really. We have just described a way in which we

will compactly describe the internal state of stress of the fluid. That's all. This approach is not limited to the mechanics of fluids, but works equally well in solid mechanics. In solid mechanics, we are often only concerned with equilibrium and thus the commonly used equilibrium statement in solid mechanics that $\nabla \cdot \mathbf{T} = 0$.

9.3.3 Meaning of $\nabla \cdot \mathbf{T}$

Above we used the definition of the stress tensor and our vector calculus to show that the net surface force acting on an arbitrary volume is the divergence of the stress tensor. We can get the same result using a free body diagram for a infinitesimal element of fluid. Consider a small cube that is dx, dy, and dz on each side. Consider the point P in the middle of the cube. To find the forces on the face of the cube in the x-direction, we must consider that the stress tensor varies in space.

The value of the stress at one point is related to the value at a nearby point through a Taylor series. Don't groan. Students always groan at Taylor series. Taylor series is your friend. Taylor series says any complicated crazy function, is just a line as long as you zoom in enough. The Taylor Series expansion of the stress tensor on the right face of the square relative to the center is,

$$T_{xx} + \frac{dx}{2}\frac{\partial T_{xx}}{\partial x}.$$

Here T_{xx} is the component of the stress tensor at the center of the cube. Likewise, the stress on the left hand face of the cube in the x-direction is,

$$T_{xx} - \frac{dx}{2}\frac{\partial T_{xx}}{\partial x}.$$

A schematic of the x component of the forces is shown for a 2D square in Figure 9.5.

Considering all the forces acting in the x-direction for a 3D cube, we

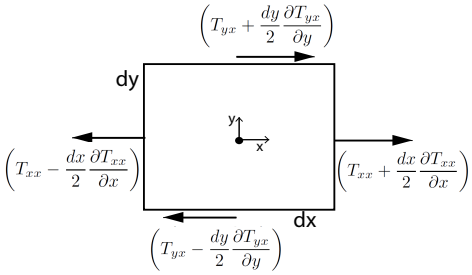


Figure 9.5 Schematic a square differential element of fluid where the stress tensor is non-constant is space. Here, we show the x component of the forces acting on the 2D differential square.

have

$$df_{sx} = \left(T_{xx} + \frac{dx}{2}\frac{\partial T_{xx}}{\partial x}\right) dydz - \left(T_{xx} - \frac{dx}{2}\frac{\partial T_{xx}}{\partial x}\right) dydz + \left(T_{yx} + \frac{dy}{2}\frac{\partial T_{yx}}{\partial y}\right) dxdz - \left(T_{yx} - \frac{dy}{2}\frac{\partial T_{yx}}{\partial y}\right) dxdz + \left(T_{zx} + \frac{dz}{2}\frac{\partial T_{zx}}{\partial z}\right) dxdy - \left(T_{zx} - \frac{dz}{2}\frac{\partial T_{zx}}{\partial z}\right) dxdy$$

Which reduces to

$$df_{sx} = \left(\frac{\partial T_{xx}}{\partial x} + \frac{\partial T_{yx}}{\partial y} + \frac{\partial T_{zx}}{\partial z}\right) dxdydz.$$

Similarly, we would find

$$df_{sy} = \left(\frac{\partial T_{xy}}{\partial x} + \frac{\partial T_{yy}}{\partial y} + \frac{\partial T_{zy}}{\partial z}\right) dxdydz,$$

$$df_{sz} = \left(\frac{\partial T_{xz}}{\partial x} + \frac{\partial T_{yz}}{\partial y} + \frac{\partial T_{zz}}{\partial z}\right) dxdydz.$$

Using vector notation,

$$\mathbf{df}_s = (\nabla \cdot \mathbf{T}) dV.$$

Thus the total force is

$$\mathbf{f}_s = \int \nabla \cdot \mathbf{T} dV.$$

and we get the same result as the previous section. The net surface

force at a point in space is given by the divergence of the stress tensor. If the stress tensor is constant in space, then there is no net force on the fluid.

9.3.4 Linear momentum, all together

Taking our definition of the rate of change of momentum and the expressions for the gravitational and surface forces we obtain,

$$\int_{V(t)} \rho \frac{D\mathbf{v}}{Dt} dV = \int_{V(t)} \rho \mathbf{g} dV + \int_{V(t)} \nabla \cdot \mathbf{T} dV. \tag{9.8}$$

Since all the integrals are volume integrals over any arbitrary volume in the fluid, the equation holds at every point,

$$\rho \frac{D\mathbf{v}}{Dt} = \rho \mathbf{g} + \nabla \cdot \mathbf{T}. \tag{9.9}$$

While this looks like a beautifully simple equation, there is a serious problem. There are too many unknowns. The equation is actually three equations for the 3 unknown components of the velocity vector. However, there are 9 components of the stress tensor. If we considered conservation of angular momentum (we will skip the derivation) we would find that the stress tensor must be symmetric. Thus there are only 6 unknown components of the tensor. This is a better situation but still 6 equations short.

What is needed to close the problem is the same type of relationship as Fourier's law provided when we studied heat conduction. We need a law that relates the unknown forces to the deformation of the material. This is called a constitutive law and cannot be consider more than an empirical relationship for the material of interest. The expression in terms of the stress tensor is general and does not make any assumption other than that of continuum. It works for fluids, solids, and other crazy materials. The choice of constitutive law determines whether we are working with fluids, solids, or something in between.

9.4 So what's a tensor?

In this chapter we introduced the idea of a stress tensor, which may seem a bit foreign to you, so it is worth emphasizing a few points again.

A scalar is used for representing things like temperature or pressure which just have a single value at a point. Temperature is not directional, it's just a number. A vector is used for representing things like velocity or heat flux because the quantity has magnitude and direction. It matters not just what the velocity is, but what direction the flow is going. We describe vectors in an orthogonal basis, i.e. $\mathbf{i}, \mathbf{j}, \mathbf{k}$.

The tensor is used to represent stress because with stress it matters which direction the force is applied as well as whether that force is a shear or tensile force. So we need 9 numbers to describe the three components of stress which can act on three faces of the material. Just like with vectors, we can project the state of stress onto any other surface from the stress tensor, but we describe it on the three orthogonal faces with their normal vectors aligned with the unit vectors. The stress tensor, represented as a 3x3 matrix, is just a way of describing the physical state of stress of the material.

Just like vectors are useful for describing a host of physical quantities. Tensors can be used for things other than the state of stress, though the most prevalent 3x3 tensors come from describing stress and deformations of materials such as fluids and solids.

10

The Navier Stokes equations

So far, for a continuum we have conservation of mass and momentum expressed as,

$$\frac{D\rho}{Dt} = -\rho \nabla \cdot \mathbf{v},$$

$$\rho \frac{D\mathbf{v}}{Dt} = \rho \mathbf{g} + \nabla \cdot \mathbf{T}.$$

These equations say nothing because we have not defined the stress tensor, $\mathbf{T}$. Defining the stress tensor requires knowledge of the material. Its definition is essentially empirical or comes from a molecular model of the material. The stress tensor is not a fundamental law of nature. The situation is just like the case of heat conduction where our fundamental conservation law was written in terms of the heat flux vector and Fourier's law was needed to close the problem. The relationship which defines the stress tensor is our constitutive law.

There are some constraints on what form the tensor must take. For example, we stated in the previous chapter (without proof) that it is symmetric due to conservation of angular momentum. It also can only take limited forms if we want its definition to remain constant under a change of coordinate systems (we always want this) or if we want it to be isotropic (we often, but not always want this). We'll save these topics for a more advanced course on continuum mechanics and not discuss them further.

10.1 Euler's equation

Euler's equation (named after Leonhard Euler, one of the greatest mathematicians in history) assumes a simple form for the stress tensor. Euler's equation assume that the only stress internal to the fluid is pressure. Pressure acts only normal to a surface and thus the tensor only has diagonal elements. The tensor has the simple form,

$$\mathbf{T} = \begin{bmatrix} -P & 0 & 0 \\ 0 & -P & 0 \\ 0 & 0 & -P \end{bmatrix}$$

With this stress tensor and the definition of the divergence we write all the terms out,

$$\nabla \cdot \mathbf{T} = \begin{bmatrix} \frac{\partial}{\partial x}, & \frac{\partial}{\partial y}, & \frac{\partial}{\partial z} \end{bmatrix} \begin{bmatrix} -P & 0 & 0 \\ 0 & -P & 0 \\ 0 & 0 & -P \end{bmatrix}$$

$$= -\left[\frac{\partial P}{\partial x}, \quad \frac{\partial P}{\partial y}, \quad \frac{\partial P}{\partial z}\right] = -\nabla P$$

Substituting this expression into our conservation law would yield conservation of momentum to be,

$$\rho \frac{D\mathbf{v}}{Dt} = -\nabla P + \rho \mathbf{g}.$$

Now we have 4 equations (1 mass and 3 momentum) and 5 unknowns, the density, velocity vector, and the pressure. This is a bit better but still a problem. The final equation to close the problem must come from thermodynamics, which relates temperature, pressure, and density. To completely define the problem we would need to introduce conservation of energy to our formulation, which we consider energy later.

However, if the flow is incompressible, we do not need to consider energy or thermodynamics. For incompressible flow, the density is constant and taken as a material property. For an incompressible flow, Euler's equations reduce to,

$$\nabla \cdot \mathbf{v} = 0, \tag{10.1}$$

$$\rho \frac{D\mathbf{v}}{Dt} = -\nabla P + \rho \mathbf{g}. \tag{10.2}$$

where the density is taken as a known and constant material property. Finally, we have 4 equations and 4 unknowns (velocity vector and pressure) and a closed set of equations. To solve the equations we need initial and boundary conditions. The commonly used boundary equation at the interface between a fluid and solid in Euler's equation is that the velocity normal to the surface must be zero; i.e. fluid cannot go into a solid surface.

The above formulation looks pretty good though there are some complexities hidden in that equation. Too bad that the Euler equations turn out not to be very useful. The reasons for Euler's equations to be of limited value are subtle and not obvious. We will discuss their failing in more detail as we progress.

The main feature of the constitutive law is that Euler's equation assumes zero viscosity. It seems in many cases the force due to viscosity should be quite small and neglecting it in the equations doesn't seem like a bad approximation. For now, you can comfort yourself in that the failings of Euler's equations to provide many predictions of practical utility stumped many smart people for many years. About 100 years spanned between Euler's development with no viscosity and the development of the Navier-Stokes equations with viscosity. It took perhaps another 50 years for to truly understand and appreciate why Euler's equations can fail as an approximation even when the viscosity is small. If some of these points seem difficult, you are in good company.

Before we get too dismissive of Euler's equations, we will see that they can be used for simple calculations, estimations, and qualitative explanations of flow behavior. So Euler's equations still have a lot of value and we will discuss them a lot in coming chapters.

Finally, there is an interesting subtlety when we made the incompressible assumption, pressure loses its meaning as a thermodynamic quantity. In an incompressible flow, information propagates infinitely fast. If I start to move, the flow adjusts everywhere instantaneously. In reality, information in a fluid propagates at the speed of sound. Thermodynamics connects the speed of sound to the thermodynamic function, $\partial P/\partial \rho$. Since the change in density, $d\rho$ is zero, then the speed of sound is infinite. However, thermodynamics is now unhappy with our definition of pressure. None of this is important for our discussion, its just interesting.

10.2 Newtonian fluid & incompressible flow

In fluid mechanics the most common constitutive relationship that accounts for viscosity is that of the Newtonian fluid. The law works well for air, water, oils and other simple fluids. It does not work well for a variety of complex fluids such as visco-elastic or polymeric liquids. The experimental observation was discussed in the opening chapter. A simple device for quantifying viscosity comprises two concentric cylinders with a very narrow gap between them. A fluid of interest is placed in the gap. The inner cylinder is held fixed while the outer one rotates at a constant angular speed, ω. The speed of the wall at the outer cylinder is simply $U = \omega R$, where R is the radius of the cylinder. If we conducted this experiment for different gap sizes d and different speeds, we would find the torque needed to spin at constant speed would follow a law,

$$\text{Torque} \propto \mu \frac{U}{d}$$

In a more general form, the Newtonian assumption relates the shear stress to the velocity gradient in the y-direction through the fluid viscosity, μ,

$$\tau = \mu \frac{\partial u}{\partial y}.$$

This is a very important concept to commit to memory: A Newtonian fluid relates stress to velocity gradients.

The general form of the Newtonian stress tensor is stated as,

$$\mathbf{T} = -\left(P + \frac{2}{3}\mu \nabla \cdot \mathbf{v}\right)\mathbf{I} + \mu\left(\nabla \mathbf{v} + \nabla \mathbf{v}^T\right). \tag{10.3}$$

Here, the superscript T denotes the transpose and $\mathbf{I}$ the identity matrix. You can start to notice a few things. First the pressure term is just like in Euler's equation, if I set the viscosity, μ, to zero then the tensor limits to that in Euler's equation. The other term added to the pressure is the proportional to the divergence of the velocity field. Recall the divergence of the velocity is a scalar. This term has to do with viscosity resisting changes in volume of the fluid. We will typically only deal with incompressible flows in this course and we can use the incompressible form,

$$\mathbf{T} = -P\mathbf{I} + \mu\left(\nabla \mathbf{v} + \nabla \mathbf{v}^T\right). \tag{10.4}$$

The terms with the velocity gradients is a little more complicated and will be the subject of much discussion later in this chapter. Since velocity is a vector, it's gradient is a tensor. The velocity gradient tensor is written out as,

$$\nabla \mathbf{v} = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} \\ \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z} \end{bmatrix}$$

By adding the velocity gradient tensor to it's transpose we obtain a symmetric tensor. If we worked through the details, we would find we could express the divergence of this stress tensor as,

$$\nabla \cdot \mathbf{T} = -\nabla P + \mu \nabla^2 \mathbf{v}.$$

Substituting into the balance of linear momentum yields,

$$\rho \left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = \rho \mathbf{g} - \nabla P + \mu \nabla^2 \mathbf{v}. \tag{10.5}$$

Coupled with mass conservation for an incompressible flow, our problem is closed;

$$\nabla \cdot \mathbf{v} = 0. \tag{10.6}$$

Equations 10.5 and 10.6 represent four equations for four unknowns; the three components of velocity and the pressure. These equations are known as the incompressible Navier Stokes equations and comprise the mathematical foundation for describing fluid flow.

Just to be clear on the notation, if we expand the momentum equation in component form for a Cartesian coordinate system we obtain,

$$\rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = \rho g_x - \frac{\partial P}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right)$$

$$\rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = \rho g_y - \frac{\partial P}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right)$$

$$\rho \left(\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = \rho g_z - \frac{\partial P}{\partial z} + \mu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right)$$

The great thing about expressing the formulation in our vector calculus notation is that we can change coordinate systems and simply look

up the operators in different coordinates. The most common alternate coordinate systems are cylindrical and spherical.

Note that for incompressible flows where the density is constant, conservation of momentum can be written as,

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v}\cdot\nabla\mathbf{v}\right) = -\nabla(P+\rho g z) + \mu\nabla^2\mathbf{v}.$$

where (without loss of generality) we aligned the coordinate system such that gravity is pointing in the negative z direction. We can introduce the dynamic pressure as $P_{\text{dyn}} = P + \rho g z$ and rewrite the equation as

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v}\cdot\nabla\mathbf{v}\right) = -\nabla P_{\text{dyn}} + \mu\nabla^2\mathbf{v},$$

The dynamic pressure is the pressure that results from fluid motion and subtracts out the hydrostatic component.

This form is convenient as it allows us to essentially ignore the effect of gravity when computing fluid motion in a constant density fluid. From the above equation we can see that if there is no fluid velocity and we are at equilibrium then $\nabla P_{\text{dyn}} = 0$. At equilibrium the dynamic pressure (pressure due to fluid flow) is constant and does not impact the flow. At equilibrium the measured pressure is given as $\nabla(P+\rho g z) = 0$, meaning the pressure is only hydrostatic. In constant density flows, gravity does not play a role in determining the flow field, but it does effect the pressure. If the density is constant then gravity acts equally on the whole fluid and thus there is no flow driven by gravity. If the density of the fluid is not constant, then we could have buoyancy driven flows.

10.2.1 So what does viscosity do?

If we look at the term in the equation that has the viscosity coefficient in front of it, we might see something reminiscent of the heat equation. In fact if we find a special class of constant density flows (which we will discuss in the next chapter) which have no spatial acceleration, and no pressure gradients, then we would have flows which are governed by the heat equation, i.e. $\frac{\partial \mathbf{v}}{\partial t} = \frac{\mu}{\rho}\nabla^2\mathbf{v}$. Since this limited form of the momentum equation looks like the heat equation, it will act in the same way. Viscosity will act to smooth out the flow and "kill" spatial

curvature in the velocity field. While the full Navier Stokes equations have other more complicated terms, we can think that viscosity will always act to smooth the velocity field and pull the velocity to some uniform or simple state - just like the heat equation.

The ratio of μ/ρ appears often that it is given its own symbol, $\nu = \mu/\rho$. This parameter is called the kinematic viscosity and has units of m^2/s. The material property, μ, is properly called the dynamic viscosity. The kinematic viscosity plays a role like the thermal diffusivity, α, or the diffusion coefficient D. In thermal problems, α, sets the rate that heat propagated over some distance; the kinematic viscosity plays a similar role. The kinematic viscosity sets the rate that momentum is transferred from one sliding fluid layer to another.

10.3 Boundary conditions

In order to solve the Navier Stokes equations, boundary conditions are needed. Boundary conditions tell you how the region of interest where your equations are valid are coupled to the rest of the universe. When there is a solid surface, we typically assume that all components of the fluid velocity equal the velocity of that solid surface. For a stationary object, the velocity vector goes to zero, $\mathbf{v} = 0$. This statement means that there is no flow through the solid surface (no normal velocity) and no-slip at the surface (no tangential velocity). The condition of no flow through the solid seems obvious. The no-slip condition, on the other hand, is not a fundamental law with a very strong theoretical backing. What boundary condition to use was a serious debate in the fluid mechanics world up to the early 1900s. Today, many people still study whether the condition is strictly true or only approximately true. Recently, there have been experiments with superhydrophobic surfaces with significant amounts of fluid slip in liquid systems. The no-slip condition can also be violated in rarefied gas flows where the length scale of the object is not that large compared to the molecules mean free path. However, for most practical cases it is experimentally observed that the no-slip condition is obeyed to a very high degree of approximation. At a free surface of liquid, we can also have a condition of no stress. However, in this course we will usually consider problems of flow around some object and thus will use the condition that $\mathbf{v} = 0$.

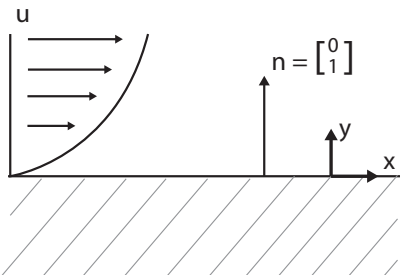


Figure 10.1 A solid surface with no slip is oriented with a normal vector in the vertical direction.

10.4 Computing stress from flows

In many cases we want to know the total force acting on an object immersed in a fluid flow. If we know the velocity field we have a way of computing the stresses at a solid surface from the stress tensor. To make our lives simpler, lets consider a 2D world. Consider the solid surface and we have aligned our coordinate system such that the normal vector for that surface points only in the y direction $\mathbf{n} = [0, 1]$; see Figure 10.1. Assume that we have through some sorcery solved the Navier Stokes equations for the full velocity field above that surface. Now we want to use that solution to calculate the force exerted on a body. In a 2D world, the Newtonian stress tensor in component form would be denoted as,

$$\mathbf{T} = -\begin{bmatrix} P & 0 \\ 0 & P \end{bmatrix} + \begin{bmatrix} 2\frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & 2\frac{\partial v}{\partial y} \end{bmatrix}.$$

The stress vector at that surface would in general be,

$$\mathbf{s} = \mathbf{n} \cdot \mathbf{T} = \begin{bmatrix} 0 & 1 \end{bmatrix} \left[-\begin{bmatrix} P & 0 \\ 0 & P \end{bmatrix} + \mu \begin{bmatrix} 2\frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & 2\frac{\partial v}{\partial y} \end{bmatrix} \right].$$

However, at our surface the vertical velocity v is zero everywhere on the surface (it is impenetrable). Therefore, $\partial v/\partial x$ is zero everywhere on the surface. Also the horizontal velocity is zero everywhere on the surface by the no slip condition, thus $\partial u/\partial x = 0$ along the surface.

Conservation of mass for an incompressible flow states that

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

At the surface, since $\partial u/\partial x = 0$ then $\partial v/\partial y = 0$. Using all the relations above which hold at the surface the problem simplifies to

$$\mathbf{s} = \mathbf{n} \cdot \mathbf{T} = \begin{bmatrix} 0 & 1 \end{bmatrix} \left[- \begin{bmatrix} P & 0 \\ 0 & P \end{bmatrix} + \mu \begin{bmatrix} 0 & \frac{\partial u}{\partial y} \\ \frac{\partial u}{\partial y} & 0 \end{bmatrix} \right].$$

Thus the stress vector at our point simplifies to,

$$\mathbf{s} = \begin{bmatrix} \mu \frac{\partial u}{\partial y}, & -P \end{bmatrix}$$

The result says that the x component of the stress is given by the horizontal velocity gradient, just as we described when defining a Newtonian fluid. This is the force that would cause a fluids viscosity to drag a solid along with the flow. The y component of the stress is given by the pressure.

10.5 Comments on kinematics

While the Navier-Stokes equation may look beautiful (or frightening), our derivation based on the stress tensor might seem arbitrary. What does the expression for the stress tensor mean and why should the stress depend upon the velocity gradient in the following way,

$$\mathbf{T} = -P\mathbf{I} + \mu \left(\nabla \mathbf{v} + \nabla \mathbf{v}^T \right). \tag{10.7}$$

In this section we will provide some interpretation of the velocity gradient and the velocity gradient tensor. The development here follows that of Kundu and Cohen (2004) quite closely.

However, we first need to emphasize again this Newtonian fluid constitutive law is not a "law". To derive it you need a molecular model of the material. This can be done for simple gases, but not many other fluids. So, in the end, we can consider this an empirical relationship and nothing more. The general form of the tensor does have some constraints placed by more fundamental laws. For example, conservation of angular momentum demands the tensor to be symmetric. Requiring the tensor to not depend on our coordinate system places restrictions on

the form. However, in the end the constitutive relationship is something that works well for many fluids, but it is not a law of nature.

10.5.1 Linear rate of strain

Consider a line element in a *steady* flow which is oriented in the x direction. A schematic is shown in Figure 10.2. Lets assume there is only an x component of the velocity field at this location. The element is dx in length. The left point, A, at time zero, $t = 0$ is at position $X_A(0)$. The right point B is at position $X_B(0)$. The positions X_A and X_B are a material points that go with the flow and are only functions of time. A short time later, dt, the material point A has gone with the flow and moved to position,

$$X_A(dt) = X_A(0) + u(X_A)dt.$$

For simplicity, the velocity field is only a function of space. The point B has moved to position,

$$X_B(dt) = X_B(0) + u(X_B)dt.$$

The distance $X_B - X_A = dx$ is meant to be infinitesimally small. The strain is defined as the change in length divided by the initial length, thus

$$\begin{aligned}
\text{Strain} &= \frac{(X_B(dt) - X_A(dt)) - dx}{dx} \\
&= \frac{X_B(0) + u(X_B)dt - X_A(0) - u(X_A)dt - dx}{dx} \\
&= \frac{(X_B(0) - X_A(0) - dx) + u(X_B)dt - u(X_A)dt}{dx} \\
&= \frac{u(X_B)dt - u(X_A)dt}{dx}
\end{aligned}$$

Therefore, in terms of a rate equation,

$$\frac{d\ \text{Strain}}{dt} = \frac{u(X_B) - u(X_A)}{dx}.$$

The velocity at point B is related to A through a Taylor Series, since the distance is assumed small. The expansion for the velocity at point $X_B = X_A + dx$ is

$$u(X_B) = u(X_A) + \frac{\partial u}{\partial x}dx.$$

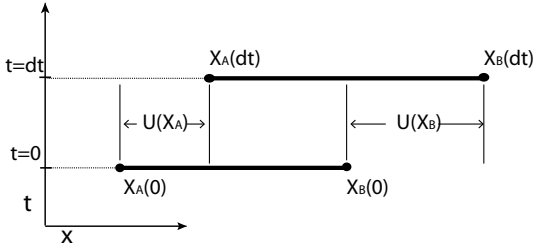


Figure 10.2 Schematic of a material line in a 1D flow. The points A and B are subjected to different velocities and thus move different distances in one instant of time, dt.

Therefore the rate of change of linear strain is related to the velocity gradient,

$$\frac{d\ \text{Strain}}{dt} = \frac{\partial u}{\partial x}.$$

The velocity gradient, $\frac{\partial u}{\partial x}$, provides the *linear strain rate* that a material line elongates or shrinks. This makes sense. Imagine you and a friend are in hot air balloons up above the earth. Since the length scale of the earth is so big we can take dx at the human scale to be infinitesimal. If the distance between you and your friend is growing, it means there are velocity gradients in the flow. Your friend, even though close by, is subjected to a different fluid velocity than you, thus you separate from each other. The rate that you separate is a measure of the flows velocity gradient.

10.5.2 Shear rate of strain

Now imagine a more complicated fluid motion in 2D. We have velocity (and velocity gradients) in both the x and y components. Now we will consider the motion of three material points, O (the origin), A, and B. As in the previous section, the location of the material points are only functions of time. However, since we are in 2D we need to consider the motion in the plane, i.e. follow (X_A, Y_A), (X_B, Y_B), and (X_O, Y_O) . The velocity field is taken as steady for simplicity of notation. Similar

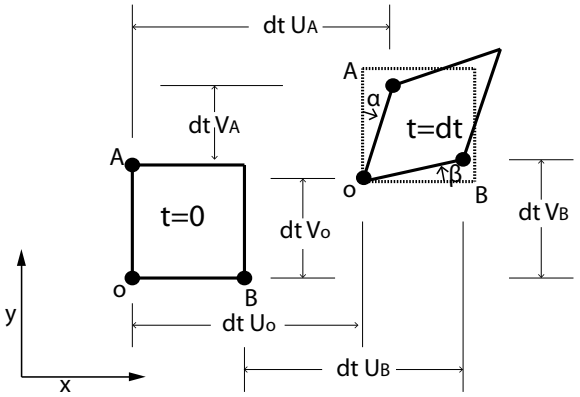


Figure 10.3 Schematic of material points O,A, and B in a 2D flow. The points are subjected to different velocities and thus move different distances in one instant of time, dt. The sides of the square are dx in length.

to the previous section, each point moves by an amount,

$$X_i(dt) = X_i(0) + u(X_i, Y_i)dt; \quad \text{and} \quad Y_i(dt) = Y_i(0) + v(X_i, Y_i)dt.$$

Here, the index i refers to point A, B or O. The above equation simply says the points move in the flow and we update their position using Euler integration in time. A schematic is shown in Figure 10.3. The three points move to new locations due to the different fluid velocities.

Using these relations and a little bit of geometry, you can find the tangent of the angles β and α in terms of the velocities at points A, B, and O as,

$$\tan(\beta) = \frac{(v_B - v_0)dt}{dx + (u_B - u_0)dt},$$

$$\tan(\alpha) = \frac{(u_A - u_0)dt}{dx + (v_A - v_0)dt}.$$

You can then use the Taylor Series to relate the velocities at point A and B to that of point O, i.e. $u_B = u_0 + \frac{\partial u}{\partial x}dx$. Substituting the

Taylor Series results and taking the limit, these geometric relationships become rate equations for the rate of change of the angles,

$$\frac{d\beta}{dt} = \frac{\partial v}{\partial x}; \qquad \frac{d\alpha}{dt} = \frac{\partial u}{\partial y}.$$

Note that in the limit of small angles which occur over small times, $\tan(\alpha) \approx \alpha$. Thus the sum of these two rates,

$$\frac{d\beta}{dt} + \frac{d\alpha}{dt} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y},$$

is the rate the angle between the two lines OA and OB close or open up. This is the *shear strain rate*.

This relationship makes sense. Hold your hands together in front of you, pointing upwards. Sliding your hands up and down relative to each other creates shear. The right hand has a different vertical velocity than the left hand. Therefore $\frac{\partial v}{\partial x}$ is not zero. Holding your hands horizontal creates velocity gradient as $\partial u/\partial y$. The shearing motion of your hands would act to deform a material held between them.

10.5.3 Rotation

Using the results from the prior section we can also ask what is the difference of the rate of change of the two angles. This difference is twice the rate of solid body rotation of the line elements at that point,

$$\frac{d\beta}{dt} - \frac{d\alpha}{dt} = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}.$$

In two dimensions, you may recognize this quantity is the same as the curl of the velocity, which is referred to in the fluid mechanics world as vorticity, ω. The curl is denoted by $\omega = \nabla \times \mathbf{v}$. In component form, the vorticity vector is

$$\omega = \nabla \times \mathbf{v} = \begin{bmatrix} \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \\ \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \\ \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \end{bmatrix}.$$

The curl is the vector cross product of the gradient operator ∇ and the velocity vector $\mathbf{v}$. In two dimensions where the velocity is constrained to the x-y plane, the vorticity only has a component normal to this

plane, the z plane.

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

The curl of the velocity is equivalent to twice the angular velocity of the fluid particle at a point. While we only made the argument in 2D, the argument holds up in a more general 3D flow.

10.5.4 Generalized to 3D

In three dimensions, the results of this section are summarized as follows. The strain rate tensor is defined as

$$\mathbf{S} = \frac{1}{2}\left(\nabla\mathbf{v} + \nabla\mathbf{v}^T\right). \tag{10.8}$$

In component form this tensor is

$$\mathbf{S} = \frac{1}{2}\begin{bmatrix} 2\frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & 2\frac{\partial v}{\partial y} & \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} & \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} & 2\frac{\partial w}{\partial z} \end{bmatrix}$$

The strain rate tensor is symmetric and provides the rate of strain of fluid elements in the different directions. The diagonal components of the tensor are the linear strain rates in the three dimensions. The off-diagonal components are the shear strain rates in the three planes; $x-y$, $x-z$, and $y-z$.

To rotation tensor is defined as

$$\mathbf{R} = \frac{1}{2}\begin{bmatrix} 0 & -\omega_z & \omega_y \\ \omega_z & 0 & \omega_x \\ -\omega_y & -\omega_x & 0 \end{bmatrix} \tag{10.9}$$

where ω are three components of the vorticity vector, $\omega = \nabla \times \mathbf{v}$. The off diagonal elements of the rotation tensor provide the amount of rotation in the three planes $x-y$, $x-z$, and $y-z$. The tensor is anti-symmetric. It is fairly easy to see that the velocity gradient is the sum of the strain rate and the rotation tensors,

$$\nabla\mathbf{v} = \mathbf{S} + \mathbf{R}.$$

Velocity gradients lead to both strain of material elements and rotation of those elements. The strain rate tensor provides the rate of strain of the material. The rotation tensor provides the amount of rotation.

Using the above tensors, the Newtonian constitutive law for an incompressible flow states that

$$\mathbf{T} = P\mathbf{I} + 2\mu\mathbf{S}.$$

The forces exerted on the fluid are proportional the strain rate, but not the rotation of the fluid.

Please note that this discussion of kinematics is quite brief and more detailed derivations can be found in the references. The main point is to demonstrate that the form of the Newtonian stress tensor is connected to the kinematics of rate of deformation of the material. The stress is related to the velocity gradients. However, the use of the symmetric strain rate tensor subtracts our solid body rotation which does not generate stress. In a Newtonian fluid only the rate of deformation generates stress.

10.6 Non-dimensionalization

Consider the Navier-Stokes equations for incompressible constant density flow,

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v}\cdot\nabla\mathbf{v}\right) = -\nabla P + \mu\nabla^2\mathbf{v},$$

$$\nabla\cdot\mathbf{v} = 0.$$

Let's say the geometry of the problem provides some characteristic length, L, and some characteristic flow velocity, U_0. The equations can be made dimensionless following the same procedure as in previous chapters. Let's define $\tilde{\mathbf{v}} = \mathbf{v}/U_0$, $[\tilde{x},\tilde{y},\tilde{z}] = [x,y,z]/L$, $\tilde{t} = tU_0/L$, and $\tilde{P} = P/P_0$. Our time scale is picked so that it takes one time unit for a fluid particle to travel the length L; i.e. our dimensionless time is $\tilde{t} = tU/L$. When making equations dimensionless, it is always important to recall that derivatives have units. The derivative, $\partial/\partial x$ has units of 1/Length, for example. Therefore, when we make our operator ∇ dimensionless, we make the substitution $\tilde{\nabla} = L\nabla$. The dimensionless version of the Laplacian is given as $\tilde{\nabla}^2 = L^2\nabla^2$. Making this change of variables would yield,

$$\frac{\rho U_0^2}{L}\left(\frac{\partial \tilde{\mathbf{v}}}{\partial \tilde{t}} + \tilde{\mathbf{v}}\cdot\tilde{\nabla}\tilde{\mathbf{v}}\right) = -\frac{P_0}{L}\tilde{\nabla}\tilde{P} + \frac{\mu U_0}{L^2}\nabla^2\tilde{\mathbf{v}},$$

$$\tilde{\nabla} \cdot \tilde{\mathbf{v}} = 0.$$

Rearranging the momentum equation and defining $P_0 \equiv \rho U_0^2$ we would obtain,

$$\left(\frac{\partial \tilde{\mathbf{v}}}{\partial \tilde{t}} + \tilde{\mathbf{v}} \cdot \tilde{\nabla} \tilde{\mathbf{v}} \right) = \tilde{\nabla} \tilde{P} + \frac{\mu}{\rho U_0 L} \nabla^2 \tilde{\mathbf{v}},$$

$$\tilde{\nabla} \cdot \tilde{\mathbf{v}} = 0.$$

Thus the problem has only one dimensionless parameter that matters, the Reynolds number,

$$\mathrm{Re} \equiv \frac{\rho U_0 L}{\mu}.$$

The Reynolds number came out in several of our problems early in this book when we discussed dimensional analysis.

10.7 The Reynolds number

From the non-dimensionalization the Reynolds number shows up as the parameter upon which the flow field depends. This parameter showed up in the first chapter where we showed that it could be used to scale all the drag coefficient and pipe flow data onto a single master curve. The Reynolds number shows up as the most important parameter in the Navier-Stokes equations. The Reynolds number is *the* parameter when it comes to determining the character of incompressible flow.

Physically, the Reynolds number is the ratio of inertia to viscosity. When the Reynolds number is large, then inertia dominates and when it is small then viscosity dominates. Often in practice high Reynolds number flows are unstable and subject to turbulence. Low Reynolds number flows are smooth. What counts as a "high" or "low" Reynolds number often depends on the situation.

The interesting thing is that since the flow only depends upon the Reynold number, a model experiment is perfectly valid if you match the Reynolds number. If I am interested in the character of the flow around a tall building, I can create a small model, match the flow in my wind tunnel to have the same Reynolds number as the thing I am interested and I will get the same flow.

Just to get a feel for what the magnitude of the Reynolds number

is, lets consider a few hypothetical cases. For water, the density is $\rho = 1000$ kg/m^3 and the viscosity is approximately $\mu = 0.001$ Ns/m^2. So lets say a person 2m tall who can swim at 1 m/s would have a Reynolds number of 2×10^6. A small organism that is 100 μm and can swim one body length per second would have a Reynolds number of 0.01. For you to match the Reynolds number of this organism you would need to swim at about the same speed (100 μm/s) in molasses!

In a pipe flow, a standard rule of thumb is that the flow will transition from laminar to turbulent at a Reynolds number of 2300. Here the Reynolds number is defined on the diameter of the pipe. To get a sense of where a Reynolds number of 2300 is - water flowing in a 1/4 inch pipe (6.35 mm) at a velocity of 1 ft/s (0.3 m/s) we would be at a Reynolds number of about 2000. The transition number is only a rule of thumb. If you try this experimentally you will find the transition number changes based on how carefully you set up the experiment. If you just hook up some tubing and push water through it without worrying about keeping everything smooth, straight, and well conditioned, you will probably observe a transition at a much lower Reynolds number, probably under 1000. If you are extremely careful and take great pains to do a perfect experiment the flow will transition at a much high Reynolds number. In 2011, the best theoretical determination of a true transition Reynolds number to sustained turbulence was determined to be Re = 2040. While a seemingly simple problem, the details are quite complex.

Reynolds original 1885 experimental apparatus survived at the University of Manchester, where he was a professor. A century later, his exact experiments were recreated with his apparatus. The experiment showed a transition Reynolds number much lower than he reported. The difference was found to be due to cars in modern day Manchester, the small amount of shaking from the streets perturbed the simple but delicate experiments.

10.8 Summary

The starting point for much of our discussion in future chapters are the Navier-Stokes equations for incompressible constant density flow,

$$\rho \left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = \rho \mathbf{g} - \nabla P + \mu \nabla^2 \mathbf{v},$$

$$\nabla \cdot \mathbf{v} = 0.$$

These equations are the formal statement of conservation of mass and momentum (i.e. Newton's laws of motion). The equations in this form assume a Newtonian fluid constitutive law which relate the instantaneous rate of strain to stresses. Not all fluids follow the Newtonian assumption but it is a good model of simple fluids such as air, water, and oil. To achieve Euler's equations simply take the limit that the viscosity is zero and remove that term from the equations. This zero viscosity limit is a funny one which we will address in a later chapter.

In the past two chapters we worked through the derivation of these equations. The discussion of the past two chapters was not as complete or detailed as can be found in many excellent textbooks. The aim of my discussion was to try to give you a sense of where the equations come from and how to think about the physical meaning of the different terms. At this point in your study, I would expect that many of the details seem at least somewhat confusing.

Of course the equations on their own don't mean much. We will need to understand the solution of these equations in order to connect to physical phenomena. Even though you now have a complete mathematical formulation for fluid flow in hand, your study (and mine) of the subject is just beginning. To move forward in our understanding of fluid flow we need to learn how to use these equations to explain phenomena that we observe everyday. As explained previously, the Navier Stokes equations are very difficult to solve, so we will need to use simple solutions to give us insight to the fundamentals and learn to use the equations to guide our qualitative thinking. There is no one better to make this point than Richard Feynman in his famous *Lectures on Physics*. Italicized words are ones I changed to switch the context from electromagnetism to fluid dynamics;

> Mathematicians, or people who have very mathematical minds, are often led astray when "studying" physics because they lose sight of the physics. They say: "Look, these differential equations - *the Navier-Stokes equations* - are all there is to *fluid dynamics*; it is admitted by the physicists *and engineers* that there is nothing which is not contained in the equations. The equations are complicated, but after all they are only mathematical equations and if I understand them mathematically inside out, I will understand the physics inside out". Only it doesn't work that way. ... They fail because the actual physical situations in the real world are so complicated that it is necessary to

have a much broader understanding of the equations. What it means to really understand an equation - that is, in more than a strictly mathematical sense - was described by Dirac. He said: "I understand what an equation means if I have a way of figuring out the characteristics of the solution without actually solving it". A physical understanding is a completely unmathematical, imprecise, and inexact thing, but absolutely necessary for a physicist *and engineer*.

My hope with the subsequent chapters is to begin to help you "understand" the Navier-Stokes equations in this imprecise and inexact sense.

11

Solutions to the Navier-Stokes equations

The Navier-Stokes equations are hard to solve. A general solution or technique for all problems does not exist. Solutions to flow problems are an active area of current research and fluid dynamics has continually driven a large number of techniques in applied mathematics going back all the way to the original work of Euler. There is no way we can cover many of the varied mathematical techniques for analyzing fluid mechanics in a first course. Going much further than we do in this text would require a stronger background in partial differential equations, a challenging topic in its own right. Computationally, the situation is perhaps equally challenging. We can use modern computational fluid dynamics (CFD) packages to solve numerous flow problems, however in many cases these solutions must be approached with caution.

But do not despair. There are some simple (and practical) cases where the Navier Stokes equations can be easily solved with hand calculations. These simple problems are insightful as they can lead to physical intuition and help connect the mathematical formulation to observable phenomena. We will show some examples in this chapter of some classical flow problems. It is important to realize that the Navier Stokes equations are non-linear equations and their solutions are not unique. If you find a solution, it might not be the one you observe in the lab. Often the simple mathematical solutions turn out to be unstable ones. While these facts might seem annoying to you, it is these facts exactly that give fluid flows some of the their beautiful patterns and makes the subject still interesting even though it has existed in its modern form for over a century.

In this chapter, we will ignore all these complexities for now and

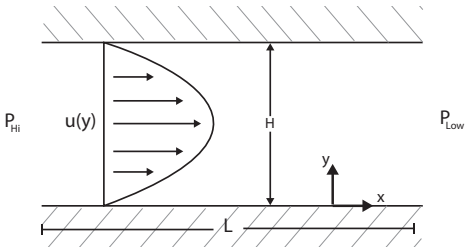


Figure 11.1 Schematic for Poiseuille flow between two parallel plates.

solve some simple problems. In all the examples that follow we will study flows where there is no acceleration and force due to viscosity will balance some driving force.

11.1 Flow between parallel plates - Poiseuille flow

The following problem is often referred to as Poiseuille flow after the French physicist and physiologist who experimentally studied the flow of liquid in narrow tubes to understand the flow of blood in the human body. We will work this problem in great detail and then use what we learned to quickly solve other related problems. We will consider here the two-dimensional version of the problem. Imagine a flow in 2D slot between solid walls with a steady, pressure driven flow. The channel height is H and we will take $y = 0$ to be the lower wall. Gravity will point in the negative y direction. We will assume that the channel is infinite in the axial x-direction. We assume the flow is not only steady in time and also flow is "steady" in the axial direction meaning that since the channel is long and the flow has no variation in the x direction.

Since the channel is infinite in x we assume that there is no reason the flow at any x location will be different than any other. Thus the axial gradients are zero; $\partial u/\partial x = 0$ and $\partial v/\partial x = 0$. Making the assumption of uniform, steady flow, we can rewrite the Navier-Stokes equations, crossing all the terms with x derivatives and time derivatives as follows,

$$\rho\left(\cancel{\frac{\partial u}{\partial t}} + u\cancel{\frac{\partial u}{\partial x}} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial P}{\partial x} + \mu\left(\cancel{\frac{\partial^2 u}{\partial x^2}} + \frac{\partial^2 u}{\partial y^2}\right)$$

$$\rho\left(\cancel{\frac{\partial v}{\partial t}}+\cancel{u\frac{\partial v}{\partial x}}+v\frac{\partial v}{\partial y}\right)=-\frac{\partial P}{\partial y}-\rho g+\mu\left(\cancel{\frac{\partial^2 v}{\partial x^2}}+\frac{\partial^2 v}{\partial y^2}\right)$$

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

Note that we keep the axial pressure gradient as the pressure drives the flow and must drop continuously as we move down the channel.

From conservation of mass, $\partial v/\partial y = 0$; an equation which states that the vertical velocity is a constant in the y direction. Since $v = 0$ at the wall, $v = 0$ everywhere. We can now cancel out all terms that have the vertical velocity, v in them;

$$\rho\left(\cancel{\frac{\partial u}{\partial t}}+\cancel{u\frac{\partial u}{\partial x}}+\cancel{v\frac{\partial u}{\partial y}}\right)=-\frac{\partial P}{\partial x}+\mu\left(\cancel{\frac{\partial^2 u}{\partial x^2}}+\frac{\partial^2 u}{\partial y^2}\right)$$

$$\rho\left(\cancel{\frac{\partial v}{\partial t}}+\cancel{u\frac{\partial v}{\partial x}}+\cancel{v\frac{\partial v}{\partial y}}\right)=-\frac{\partial P}{\partial y}-\rho g+\mu\left(\cancel{\frac{\partial^2 v}{\partial x^2}}+\cancel{\frac{\partial^2 v}{\partial y^2}}\right)$$

It makes sense that the left side of the Navier Stokes equations is zero. The left side represents the acceleration of a fluid particle. There is no acceleration in this system. There is neither acceleration due to unsteadiness (change in velocity with respect to time) nor from the perspective of a fluid particle. The channel is like a calm river whose width doesn't change; if you were on a raft you would translate at constant velocity. Neither the observer on the raft or on the banks would see any acceleration in this flow.

Thus, the momentum equations are simplified significantly to

$$\frac{\partial P}{\partial x}=\mu\frac{\partial^2 u}{\partial y^2}. \tag{11.1}$$

$$\frac{\partial P}{\partial y}=-\rho g. \tag{11.2}$$

Integration of the second equation yields,

$$P = P(x, y = 0) - \rho g y.$$

where $P(x, y = 0)$ is the pressure along the lower wall, which is a function of x only. When we take the pressure gradient in the x direction, the hydrostatic component (the term with ρg) does nothing to modify

the axial flow. Since the flow was assumed invariant in x, the only consistent solution would be that the axial pressure gradient, dP/dx, is a *constant*. Since the left side of Equation 11.1 is a constant the equation can be easily integrated twice with respect to y,

$$u(y) = \frac{dP}{dx}\frac{1}{2\mu}y^2 + C_1 y + C_2.$$

We are left with two constants of integration which are determined by the no-slip boundary conditions at both walls. For the lower wall we set $u(y = 0) = 0$. Substituting in this boundary condition states that the constant C_2 must be zero. The upper wall condition of $u(y = H) = 0$ allows us to determine C_1. The final result is,

$$u(y) = -\frac{dP}{dx}\frac{1}{2\mu}y(H - y).$$

The velocity profile follows a simple parabola which is zero at $y = 0$ and $y = H$. Recall from our discussion on heat conduction that the second derivative in space is equivalent to the curvature. Therefore, the governing equation states that the curvature of the x component of the velocity field is proportional to a constant (the pressure gradient). The parabola is the only function which has constant curvature, thus we could obtain the qualitative shape of the velocity profile without even solving the equation.

The total flow, per unit width is $Q = \int_0^H u(y)dy$. Integrating our velocity profile gives the total flow as

$$Q = -\frac{dP}{dx}\frac{H^3}{12\mu}$$

Usually we use the fact that dP/dx is the same as the total applied pressure difference divided by the total length of channel. We need to be careful with the signs. Convention would usually define a positive pressure difference, ΔP, as going from high pressure on the inlet to low pressure on the outlet. With this convention, $dP/dx = -\Delta P/L$. Using the total pressure drop notation gives,

$$\Delta P = Q\frac{12\mu L}{H^3}.$$

The expression says that pressure and flow rate are linearly related - double the pressure and you double the flow. This expression is a hydraulic version of Ohm's law ($V = iR$) where pressure difference

acts like voltage and volumetric flow rate acts like current. The number $12\mu L/H^3$ is the hydraulic resistance.

There are a few things we should check with our resistance formula. First, we can check the units to make sure they agree with what we expect and that we did not make an error. The other check is that the trends all go in the right way. It makes physical sense that the resistance increases with viscosity, channel length, and with a decrease in the channel height. It is always good to look at a result and see if you believe the basic facts about that result before proceeding.

Note that the mean flow velocity is $\bar{u} = Q/H$,

$$\bar{u} = \frac{\Delta P H^2}{12 L \mu}$$

and the maximum flow velocity (along the centerline) is found by evaluating the velocity profile at $y = H/2$,

$$u\left(y = \frac{H}{2}\right) = \frac{\Delta P H^2}{8 L \mu} = \frac{3}{2}\bar{u}.$$

Since there is no acceleration in this flow we can check our result with a simple force balance If we draw a box (control volume) the net force (per unit depth into the page) exerted on the box due to pressure is

$$F_P = \Delta P H.$$

This force must be balanced by the shear stress exerted at the two walls. Recall the method of finding the stress on a surface discussed in the chapter on the Navier-Stokes equations. The shear stress acting tangential to the wall is given by $\mathbf{n}\cdot\mathbf{T}$ where $\mathbf{T}$ is the total stress tensor. The normal vector for the lower wall is $\mathbf{n} = [0 \;\; 1]$. The stress tensor at the wall for this simplified flow is

$$\mathbf{T} = -P\mathbf{I} + \mu\left(\nabla\mathbf{v} + \nabla\mathbf{v}^T\right) = \begin{bmatrix} -P & \mu\frac{\partial u}{\partial y} \\ \mu\frac{\partial u}{\partial y} & -P \end{bmatrix}.$$

Thus the stress vector is,

$$\mathbf{s} = \mathbf{n}\cdot\mathbf{T} = \begin{bmatrix} \mu\frac{\partial u}{\partial y} \\ -P \end{bmatrix}$$

The tangential stress is proportional to the velocity gradient $\partial u/\partial y$

which for our velocity field is,

$$\frac{\partial u}{\partial y} = \frac{\Delta P}{L}\frac{1}{2\mu}(H - 2y).$$

The tangential stress, or x-component of the stress vector, acting on the lower wall is then

$$s_x(y = 0) = \mu \left.\frac{\partial u}{\partial y}\right|_{y=0} = \frac{\Delta P H}{2L}.$$

By symmetry we would expect the stress on the upper wall to be the same. If you carry through the operation as we did above you only need to be careful with the signs. The sign of the slope of the velocity field is reversed at the upper wall, but the sign of the normal vector is also reversed, thus giving a force in the same direction as the lower wall. To get the force, the tangential stress is integrated over the length of the channel, which is just the constant shear stress multiplied by the length of channel. The total shear force per unit depth into the page, F_s, is,

$$F_s = (s_x(y = 0) + s_x(y = H))L = F_P = \Delta P H.$$

The total shear force equals the total pressure force.

Could we have estimated our result from the start without actually solving the problem? We can actually get a good estimate using the force balance. The applied pressure force was given simply above. The shear stress at the surface is $\tau = \mu \partial u/\partial y$. If we did not solve the equation, we could have guessed that the velocity gradient would be of the same order of magnitude as the flow velocity divided by half the channel height; namely $\tau = \mu \partial u/\partial y \sim 2\mu U/H$. The total shear stress exerted on *both* walls is,

$$F_s \sim 4\mu \frac{U}{H} L.$$

Equating these two forces would yield an estimate of the maximum flow velocity as

$$u_{\max} \sim \frac{\Delta P H^2}{4\mu L}.$$

which is not the precise formula but has the right scaling with the parameters. A simple estimate in a simple problem seems useless (and perhaps it is), but the same type of scaling arguments can be used in

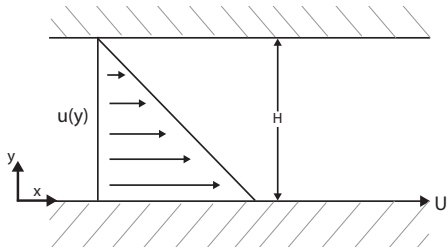


Figure 11.2 Schematic for Couette flow between two parallel plates.

more complicated problems to get a handle on the form of the answer, the basic scaling, or the order of magnitude. It is very common practice to guess that the terms in the partial differential equation can be approximated in simple ways just to get a handle on what the answer might look like.

While the parabolic velocity profile is accurate and can be realized in practice, the theoretical solution between parallel plates is only observed when the Reynolds number is less than 1400 (or thereabouts). If the Reynolds number is large, the flow becomes unstable and turbulent. The simple laminar flow solution is lost. Instability is an issue we will discuss later.

11.2 Flow driven by a wall - Couette flow

Another simple case is where the flow is driven by a moving wall, known as Couette flow. We assume a 2D channel in cartesian coordinates. The channel goes from $0 < y < H$. Just like the previous example, we will assume the channel is infinite in the x direction and is at steady state (both with respect to time and distance along the channel). This case can be realized in practice by placing fluid in a thin gap between concentric cylinders and rotating the cylinders with respect to each other. The fact that we use cartesian coordinates is akin to assuming a flat earth (i.e. the curvature is so large compared to the length scale across the channel). In this case we will assume that gravity acts normal the $x - y$ plane, and thus has no role in this problem.

We can write the full Navier-Stokes just as in the Poiseuille flow example and cross out all the time derivatives and x derivatives. We can

then apply conservation of mass to tell us there is no vertical velocity, v. Crossing out all these terms again yields the x and y momentum equations as in Equations 11.1 and 11.2,

$$\frac{\partial P}{\partial x} = \mu \frac{\partial^2 u}{\partial y^2}.$$

$$\frac{\partial P}{\partial y} = 0.$$

However, in this case the x momentum equation simplifies further as there is no applied pressure gradient. In the example of the concentric cylinders there could be no pressure gradient around the closed loop of the fluid gap, otherwise the pressure would be discontinuous as we went around the circle. For Couette flow, the x momentum equation simplifies to,

$$0 = \frac{\partial^2 u}{\partial y^2}.$$

The curvature of the velocity field is zero and thus we expect a linear velocity profile. Integrating this expression twice with respect to y yields,

$$u(y) = C_1 y + C_2.$$

The two boundary conditions used to determine the two constants of integration are $u(y = 0) = U$ and $u(y = H) = 0$. Applying these conditions to get C_1 and C_2, the full velocity field is,

$$u(y) = U \frac{H - y}{H}.$$

The velocity varies linearly between the two plates. The total flow rate between the plates is computed as

$$Q = \int_0^H u(y) dy = \frac{UH}{2}.$$

That's all there is to it.

Note the analogy with steady state heat conduction. The equation is the same as we found in one-dimensional heat conduction at steady state. Just as in steady heat conduction, the Couette flow equation states that the velocity field has no curvature. The only possible solution to this equation is a linear velocity profile.

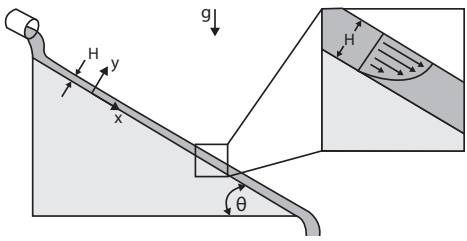


Figure 11.3 Schematic for gravity driven flow down a ramp.

11.3 Flow down a ramp

Another variation of the class of problems in simple plane parallel flow is that of a thin liquid film flowing down a ramp. Here, the ramp is inclined at an angle θ and the flow is driven by gravity. Fluid is poured in continuously at the top of the ramp. The upper surface of the fluid film is exposed to the air. We will make the same assumptions as before that the flow is steady and not varying with distance down the ramp. We orient the coordinate system with the inclination of the ramp such that the y direction goes across the film of liquid, and x goes down the ramp. The surface of the ramp will be $y = 0$ and the surface of the film will be $y = H$.

The development of the basic equations is identical to what we have seen. The fluid acceleration is zero, the y component of the velocity is zero, and all gradients of the flow velocity with respect to x are zero. Following the same procedure as before, we obtain for the x and y momentum,

$$\frac{\partial P}{\partial x} = \rho g \sin(\theta) + \mu \frac{\partial^2 u}{\partial y^2}.$$

$$\frac{\partial P}{\partial y} = -\rho g \cos(\theta).$$

Integrating the second equation across the thickness of the film and using the boundary condition that the pressure at the surface of the film is that of the air around, P_∞, we obtain the pressure everywhere.

$$P = P_\infty + \rho g \cos(\theta)(H - y)$$

The pressure does not depend on x, thus the x momentum equation is,

$$-\rho g \sin(\theta) = \mu \frac{\partial^2 u}{\partial y^2}.$$

Integrating twice yields (since the left side is constant)

$$u(y) = -\frac{\rho g \sin(\theta)}{2\mu} y^2 + C_1 y + C_2.$$

The two constants of integration are found from the boundary condition. At the surface of the solid ramp, the no-slip condition holds, $u(y = 0) = 0$. Applying this boundary condition shows that $C_2 = 0$. Therefore,

$$u(y) = -\frac{\rho g \sin(\theta)}{2\mu} y^2 + C_1 y.$$

The second boundary condition is that there is no shear stress at the free surface of the liquid. Nothing is there (other than a little air) to exert a force on the upper surface of the free fluid film. Thus we need to set $\partial u/\partial y = 0$ at the upper surface $y = H$. Applying this condition yields,

$$u = \frac{\rho g \sin(\theta)}{\mu} y \left(H - \frac{y}{2} \right).$$

You should always confirm at the end that the solution satisfies the equation and the boundary conditions.

Also note that the solution here is half a parabola - or essentially one half of the flow between two solid plates. The driving force in Poiseuille flow is pressure whereas here it is gravity. Otherwise, the solutions look very similar.

11.4 Combined Poiseuille and Couette flow

In cases where there is no fluid acceleration (or we can neglect it) the Navier Stokes equations become linear equations. Linear equations are wonderful since we can use the useful idea of superposition to construct the solution to more complex problems. We can take solutions to simple problems and add them up in clever ways to construct a new solution. Rarely in fluid mechanics are we afforded the opportunity to use superposition.

One example where we can use superposition is imagine a case where we have flow in a two-dimensional gap but that the flow is driven by a pressure gradient *and* a sliding wall. Let's take the lower wall moving at velocity U. For this problem, let's ignore gravity and assume it is pointing out of the plane. If we started with the Navier Stokes equations in two dimensions, we would cross out all the neglected terms exactly as we did in the section on Poiseuille flow arriving again at Equations 11.1 and 11.2. When we integrate the resulting equation, nothing changes,

$$u(y) = \frac{dP}{dx}\frac{1}{2\mu}y^2 + C_1 y + C_2.$$

To find the constants of integration we would need to apply the boundary conditions, $u(y = 0) = U$ and $u(y = H) = 0$. Applying the first condition yields,

$$u(y = 0) = U = C_2.$$

Applying the second condition yields,

$$u(y = H) = 0 = \frac{dP}{dx}\frac{1}{2\mu}H^2 + C_1 H + U \;\rightarrow\; C_1 = -\frac{U}{H} - \frac{dP}{dx}\frac{1}{2\mu}H.$$

The flow field is then given as

$$u(y) = -\frac{dP}{dx}\frac{1}{2\mu}y(H - y) + U\frac{H - y}{H}.$$

The total flow rate

$$Q = \int_0^H u(y) = -\frac{dP}{dx}\frac{H^3}{12\mu} + \frac{UH}{2}.$$

If you look at the velocity profile and total flow rate derived for Poiseuille and Couette flow independently, you will see the above result is just the superposition of the two solutions. We could have guessed this result from the start since under our assumptions the equations are linear. When you have linear equations you can compose a solution to a problem as the sum of other solutions. Superposition is a useful technique for solving more complex flow problems as we will demonstrate in a few of the coming examples.

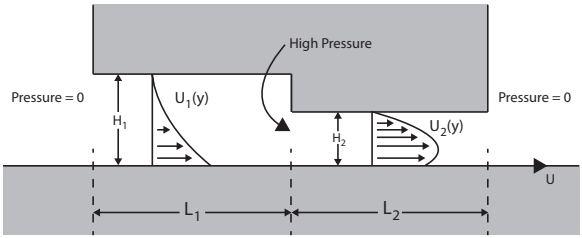


Figure 11.4 Schematic for sliding block moving to the left over a stationary surface. In the reference frame of the block the lower wall appears to move to the right at constant velocity. While the gap size is exaggerated here, the solution assumes the gap is very thin relative to the length of the block.

11.5 Slider bearing

We can use the solutions derived in simple situations to piece together what will happen in more complex situations. As an example, let's consider the motion of two solid surfaces sliding past each other as in Couette flow. However, in this case the upper surface will have a step change in the height of the gap. In Figure 11.4 we take the upper block of length $L_1 + L_2$ moving to the left. We must realize that in the reference frame of the stationary wall, the problem is unsteady. If we observe the flow from the moving block the flow appears steady in time. Thus we will take the perspective that rather than the object moving to the left, the lower flat wall is moving to the right as shown in Figure 11.4.

We will assume the gap height is small relative to the length such that we can ignore effects that might occur around the step. We will assume the flow is essentially uniform in the x-direction other than a small local region around the step. We might be tempted to think that the flow would be a simple Couette flow in the two regions. However, if that were true then the flow rate in the two regions (see the Couette flow derivation) would be $Q_1 = UH_1/2$ and $Q_2 = UH_2/2$. A linear velocity profile is not plausible since the total flow rate would be different in the two regions. Since the flow is incompressible, the total volumetric flow rate in each section under the slider must be the same. Thus, the manner in which the problem rectifies itself is that a pressure develops in the thin gap to push fluid out of the gap as shown schematically in

Figure 11.4. If the pressure is high at the step then the flow rate will be enhanced in region 2 and decreased in region 1.

The velocity profile in each region can be thought of as the superposition of Couette and Poiseuille flow. Using the results derived previously, the total flow rate in region 1 will be

$$Q_1 = -\left.\frac{dP}{dx}\right|_1 \frac{H_1^3}{12\mu} + \frac{UH_1}{2},$$

and similarly for region 2. Overall conservation of mass would state that $Q_1 = Q_2$,

$$-\left.\frac{dP}{dx}\right|_1 \frac{H_1^3}{12\mu} + \frac{UH_1}{2} = -\left.\frac{dP}{dx}\right|_2 \frac{H_2^3}{12\mu} + \frac{UH_2}{2}.$$

The expression gives us the relationship between the two pressure gradients in regions 1 and 2. To close the problem, we need additional information. We need to know what is the overall applied pressure across the entire fluid gap. Let's assume the upper slider is open to the fluid through which it moves. Therefore, the pressure at the two ends would be the same and there is no overall applied pressure. Since only pressure differences matter, we can take the far left and right ends of the slider to be zero pressure. If P is the pressure at the step then the pressure gradient in region 1 would be P/L_1 and the pressure gradient in region two is $-P/L_2$. The difference in sign is because the pressure rises from 0 to P in region 1 and falls from P to 0 in region 2. We now have enough information to solve for the pressure at the step, P,

$$P\left(\frac{1}{L_2} + \frac{1}{L_1}\frac{H_1^3}{H_2^3}\right) = \frac{6\mu U}{H_2^3}(H_1 - H_2).$$

Notice that when $H_1 = H_2$ the pressure is zero and we are back to Couette flow. Also notice that the pressure under the slider is positive when $H_1 > H_2$ and negative when the situation is reversed.

In order to understand the result a little easier, lets explore the case where $L_1 = L_2 = L/2$ and $H_2 = H_1/2$. Substituting in these parameters yields a pressure at the step of,

$$P = \frac{4\mu UL}{3H^2}.$$

The pressure grows as the gap gets smaller. The positive pressure in the gap (when $H_1 > H_2$) provides a lift force on the sliding object.

Since the pressure grows as the layer is squeezed, the fluid can prevent the two solids from coming into contact. The basic effect is used in hydraulic bearings where the viscous fluid forces replaces those of solid-solid contact.

Recall from the Poiseuille flow example that the stress acting on the solid wall is,

$$\mathbf{s} = \mathbf{n} \cdot \mathbf{T} = \begin{bmatrix} \mu \frac{\partial u}{\partial y} \\ -P \end{bmatrix}.$$

The total normal load that could be supported by the pressure in the lubrication layer is the integral of the pressure (the y component of the stress) under the slider. Since the pressure gradient is a constant, the pressure varies linearly with x and the total normal force per unit width would be,

$$F_N = \frac{4\mu U L^2}{3H^2}.$$

The force is directed upward due to the high pressure in the gap and thus there is a lift force exerted on the block.

The tangential force is that required to drag the block through the fluid. The tangential force is found by evaluating the shear stress $\tau = \mu \partial u / \partial y$ at the wall and then integrating over the length. The shear stress in region one is

$$\tau_1 = \mu \left. \frac{\partial u}{\partial y} \right|_{y=0} = -\frac{PH}{L} - \frac{U\mu}{H},$$

Substituting for P gives,

$$\tau_1 = -\frac{4\mu U}{3H} - \frac{U\mu}{H} = -\frac{7\mu U}{3H}.$$

Likewise for region 2,

$$\tau_2 = \mu \left. \frac{\partial u}{\partial y} \right|_{y=0} = \frac{P}{2L} - \frac{2U\mu}{H} = -\frac{\mu U}{3H},$$

The total tangential force per unit width is then $(\tau_1 L_1 + \tau_2 L_2)$, or

$$F_T = -\frac{8\mu U L}{3H}.$$

A variation of this problem is a classic one where instead of the step change in the height, a smooth upper surface (with no step in height)

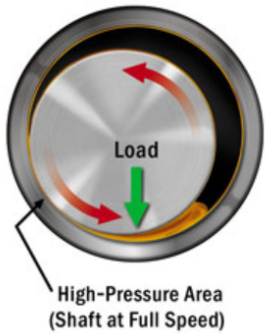


Figure 11.5 Schematic of the equilibrium position of a shaft in a lubricated bushing. From Machinery Lubrication - a trade publication (www.machinerylubrication.com).

slides at an angle relative to the lower one. This problem was solved by Reynolds (yes, the same one) shortly after his work on pipe flow and the transition from laminar to turbulent flow. The basic behavior Reynolds found is comparable to the case here, it just requires a little more analysis. When the upper surface angles upward (same as the incoming section is thicker than the trailing one) a high pressure is found in the gap which provides a lift force. This lift force can keep the upper block from contacting the lower surface. As the gap gets thinner, the pressure and thus the lift force increases so there is a stabilizing tendency. You can observe this effect by tossing playing cards across a table. Incline the card upward and give it a good flick and it will slide a long distance. You probably know this if you play a lot of poker (or other card games). If you try this experiment, you can also try to punch holes in the card such that the pressure can't build up in the gap. The cards with holes will come to a quick halt when you try to slide them.

A shaft rotating through a bushing is an important component of one of our most useful inventions; the wheel. It has been long known that lubricating the gap between the shaft and bushing with oil or other viscous fluid can dramatically reduce friction. The problem we worked in this section is the beginning of the study of this classic journal bearing problem. If there is load on the shaft, there will be a tendency for the shaft to move and contact the bushing. However, when the shaft

is rotating, we now have flow in a thin gap of nonuniform height. A lubrication pressure will build in the gap and push the shaft away from the bushing. Since the smaller the gap the higher pressure, eventually the system will come to an equilibrium position where the forces balances and the shaft is held off the bushing by the lubrication pressure. It turns out that the lubrication force is not aligned with the load, so the shaft will sit off center in the bushing as shown schematically in Figure 11.5. The principle is also important in thrust bearings.

11.6 Impulsively started Couette flow

Now let's consider a transient flow problem in the $x-y$ plane. Imagine the Couette flow problem where initially both plates and the fluid are at rest. At $t = 0$, the plate is impulsively started to move with a constant velocity U and we wish to know the velocity field at any time, t. There is no applied axial pressure gradient in this example.

We can start with the reduction of the Navier-Stokes equations, which will proceed along the same lines as all the previous examples. Since the plate is infinite in x there is no reason to think that the flow at any x location should be different than any other. Thus all terms with $\partial/\partial x$ are set to zero. Applying this reduction to conservation of mass yields,

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

Therefore, $\partial v/\partial y = 0$, and the vertical velocity is a constant in the y direction. Since $v = 0$ at the plate, $v = 0$ everywhere. *Even though the flow is transient in time, the argument from conservation of mass about why the vertical velocity is zero remains the same since conservation of mass is satisfied instantaneously in an incompressible flow.*

We can now turn to the x component of the momentum in the Navier-Stokes equations and we can discard terms that are multiplied by v or have x gradients. The complete x-momentum equation with discarded terms is,

$$\rho\left(\frac{\partial u}{\partial t} + u\cancel{\frac{\partial u}{\partial x}} + v\cancel{\frac{\partial u}{\partial y}}\right) = \cancel{\rho g_x} - \cancel{\frac{\partial P}{\partial x}} + \mu\left(\cancel{\frac{\partial^2 u}{\partial x^2}} + \frac{\partial^2 u}{\partial y^2}\right).$$

The Navier-Stokes is simplified significantly to

$$\rho \frac{\partial u}{\partial t} = \mu \frac{\partial^2 u}{\partial y^2}.$$

The reduction for the Navier-Stokes is precisely as we had for steady Couette flow, only now we have retained the transient term.

Dividing by the density and recalling the μ/ρ is the kinematic viscosity ν, we have something that looks suspiciously like the 1D heat equation,

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2}.$$

The kinematic viscosity plays the same role in this equation as thermal diffusivity in the heat equation. We expect that the behavior would be the same as applying an instantaneous change in temperature to one side of a block of material while holding the other side cold. In this flow case, we are talking about diffusion of momentum instead of diffusion of thermal energy. The kinematic viscosity ν has the same units as the thermal diffusivity.

Let's make this equation dimensionless. Setting a scale for velocity and length seems obvious; $\tilde{u} = u/U$, $\tilde{y} = y/H$. However, what should the time scale be? Using an arbitrary scale $\tilde{t} = t/t_0$ would yield,

$$\frac{\partial \tilde{u}}{\partial \tilde{t}} = \frac{\nu t_0}{H^2} \frac{\partial^2 \tilde{u}}{\partial \tilde{y}^2}.$$

So clearly the "right" choice is to set $t_o \equiv H^2/\nu$ which yields the problem,

$$\frac{\partial \tilde{u}}{\partial \tilde{t}} = \frac{\partial^2 \tilde{u}}{\partial \tilde{y}^2}.$$

with initial condition $\tilde{u}(\tilde{t} = 0, \tilde{y}) = 0$, and boundary conditions

$$\tilde{u}(\tilde{t}, \tilde{y} = 0) = 1, \quad \text{and} \quad \tilde{u}(\tilde{t}, \tilde{y} = 1) = 0.$$

Therefore the solution at steady state will be $\tilde{u}(\tilde{y}) = 1 - \tilde{y}$. The system will approach this equilibrium state with a time scale on the order of H^2/ν. The dimensionless formulation of this problem is exactly as we found for similar problems in heat conduction.

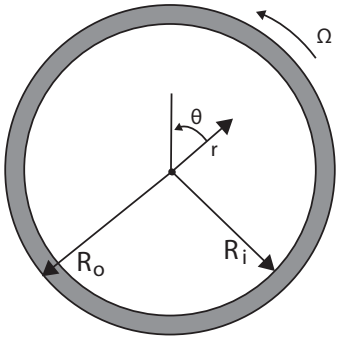


Figure 11.6 Schematic for Couette flow between two concentric cylinders. The outer cylinder with radius R_o rotates with angular velocity Ω.

11.7 Cylindrical Couette flow

Previously we solved the flow between two sliding plates. This flow can be approximated in the lab by spinning two concentric cylinders relative to each other. Rather than making the cartesian assumption for a thin gap, let's solve the problem in cylindrical coordinates (r, θ, z). The nice thing about using the Navier-Stokes equations written in vector form is that we can keep the equations the same and easily transform to another coordinate system such as cylindrical or spherical. We will not derive the equations in different coordinate system but they can be found in a number of references and easily looked up. Lets take a outer cylinder of radius R_o rotating at constant angular velocity of Ω. The inner cylinder is stationary with radius R_i. The fluid is infinite in the z direction, and gravity acts in the z direction. We will assume that the solution is axisymmetric (there is no variation in the θ direction) and that there is no fluid motion in the z direction. This seems reasonable to start. We will use the notation that u_r is the radial component of velocity and v_θ is the θ velocity.

The momentum equations in the r and θ directions are stated below (by looking up the vector operators in cylindrical coordinates). To simplify matters, we ignore gradients in the z direction and assume that everything is constant in z. The momentum and mass equation are

stated where we cancel unsteady terms and ones that disappear due to the axisymmetric assumption. The equations with discarded terms are for the r momentum,

$$\rho\left(\cancel{\frac{\partial u_r}{\partial t}} + u_r\frac{\partial u_r}{\partial r} + \cancel{\frac{v_\theta}{r}\frac{\partial u_r}{\partial \theta}} - \frac{v_\theta^2}{r}\right) = -\frac{\partial P}{\partial r} +$$

$$\mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial r u_r}{\partial r}\right) + \cancel{\frac{1}{r^2}\frac{\partial^2 u_r}{\partial \theta^2}} - \cancel{\frac{2}{r^2}\frac{\partial u_\theta}{\partial \theta}}\right),$$

θ momentum,

$$\rho\left(\cancel{\frac{\partial v_\theta}{\partial t}} + u_r\frac{\partial v_\theta}{\partial r} + \cancel{\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}} + \frac{u_r v_\theta}{r}\right) = \cancel{-\frac{1}{r}\frac{\partial P}{\partial \theta}} +$$

$$\mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial r v_\theta}{\partial r}\right) + \cancel{\frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2}} + \frac{2}{r^2}\frac{\partial u_r}{\partial \theta}\right),$$

and conservation of mass,

$$\frac{1}{r}\frac{\partial r u_r}{\partial r} + \cancel{\frac{1}{r}\frac{\partial v_\theta}{\partial \theta}} = 0.$$

Integration of conservation of mass tells us the ru_r =Constant. Since the radial velocity at the surface of the cylinder is zero, the radial velocity is zero everywhere. This further simplifies the momentum equations to

$$\rho\left(\cancel{\frac{\partial u_r}{\partial t}} + \cancel{u_r\frac{\partial u_r}{\partial r}} + \cancel{\frac{v_\theta}{r}\frac{\partial u_r}{\partial \theta}} - \frac{v_\theta^2}{r}\right) = -\frac{\partial P}{\partial r} +$$

$$\mu\left(\cancel{\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial r u_r}{\partial r}\right)} + \cancel{\frac{1}{r^2}\frac{\partial^2 u_r}{\partial \theta^2}} - \cancel{\frac{2}{r^2}\frac{\partial u_\theta}{\partial \theta}}\right),$$

θ momentum,

$$\rho\left(\cancel{\frac{\partial v_\theta}{\partial t}} + \cancel{u_r\frac{\partial v_\theta}{\partial r}} + \cancel{\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}} + \cancel{\frac{u_r v_\theta}{r}}\right) = \cancel{-\frac{1}{r}\frac{\partial P}{\partial \theta}} +$$

$$\mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial r v_\theta}{\partial r}\right) + \cancel{\frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2}} + \cancel{\frac{2}{r^2}\frac{\partial u_r}{\partial \theta}}\right),$$

The equations are significantly simplified just as in the other examples of this chapter. One difference however, is that there is one acceleration term that shows up in the radial momentum balance. This is the centripetal acceleration. This acceleration is balanced by the radial pressure gradient. The radial pressure is analogous to the case where you swing an object tied to the string in a circle above your head and the tension in the string balances the object's acceleration. The equations we will solve are the radial momentum balance,

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

and the θ momentum,

$$0 = \mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial r v_\theta}{\partial r}\right)\right)$$

Integrating the θ momentum equation once yields,

$$\frac{1}{r}\frac{\partial r v_\theta}{\partial r} = C_1$$

which can be integrated again,

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

Solving for the constants of integration by applying the boundary condition at the inner and outer cylinder gives the velocity field as,

$$v_\theta = \Omega R_o \frac{\frac{r}{R_i} - \frac{R_i}{r}}{\frac{R_o}{R_i} - \frac{R_i}{R_o}}$$

It is easy to confirm that this solution satisfies the boundary conditions. If you plot the solution for cases where the inner cylinder has a radius which is 90 percent of the outer cylinder, you will notice little difference between the true solution and the linear "flat earth" approximation.

The flow between concentric cylinders is a classic problem called Taylor-Couette flow. The general Taylor-Couette problem considers the ability to spin the inner and outer cylinder wall independently. You should check out some movies of this flow on youtube. The simple flow with the velocity field above can be only observed in some regions of parameter space. As the relative rotation speeds of the cylinders are changed, the flow can transition through a number of complicated

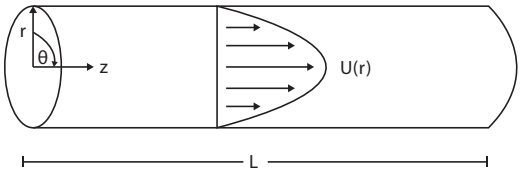


Figure 11.7 Schematic for pressure driven Poiseuille flow in a pipe.

shapes and regular but complicated patterns. This behavior is indicative of the non-linear behavior of the Navier-Stokes equations.

11.8 Poiseuille flow in a pipe

Analysis of flow in a circular pipe of radius R follows the same logic as we discussed for Poiseuille flow in a slot. However, since in the laboratory or in application it is much easier and more common to have flow in a circular pipe this is an important example. As in the slot example, we will assume that the pipe is very long and we are interested in the steady state behavior along the length of the pipe. We will assume that we are only interested in flows that are axisymmetric. If we write the Navier-Stokes in cylindrical coordinates, we can proceed by eliminating terms based on the axisymmetric assumption and that the gradients in the axial direction are zero. We will use the notation that u_r is the radial component of velocity, u_θ is the θ velocity, and u_z is the axial velocity.

Writing conservation of mass in cylindrical coordinates and discarding gradients in θ and z yields.

$$\frac{1}{r}\frac{\partial r u_r}{\partial r} + \cancel{\frac{1}{r}\frac{\partial u_\theta}{\partial \theta}} + \cancel{\frac{\partial u_z}{\partial z}} = 0.$$

Conservation of mass states that ru_r is a constant. Since the radial component of the velocity is zero at the wall $r = R$, it is zero everywhere.

Since there is no radial or swirling (θ) velocity those components of the momentum equation will all be zero. We only need to consider the

axial momentum equation,

$$\rho\left(\cancel{\frac{\partial u_z}{\partial t}}+\cancel{u_r\frac{\partial u_z}{\partial r}}+\cancel{\frac{u_\theta}{r}\frac{\partial u_z}{\partial \theta}}+\cancel{u_z\frac{\partial u_z}{\partial z}}\right)=-\frac{\partial P}{\partial z}+$$

$$\mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right)+\cancel{\frac{1}{r^2}\frac{\partial^2 u_z}{\partial \theta^2}}+\cancel{\frac{\partial^2 u_z}{\partial z^2}}\right),$$

Since there is only one component of the velocity we drop the subscript on the u_z and just use u to denote the axial velocity for simplicity. The final reduced momentum equation is analogous to Equation 11.1 in cylindrical coordinates,

$$\frac{dP}{dz}=\mu\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u}{\partial r}\right). \tag{11.3}$$

Integrating twice gives

$$u(r)=\frac{1}{4\mu}\frac{dP}{dz}r^2+C_1\ln(r)+C_2$$

Since we don't want a singularity at $r=0$, $C_1=0$. Applying the no-slip boundary condition that $u(r=R)=0$ gives,

$$u(r)=\frac{dP}{dz}\frac{1}{4\mu}(r^2-R^2).$$

The total flow rate, Q, is found by

$$Q=\int_0^R 2\pi r u(r)dr=2\pi\frac{dP}{dz}\frac{1}{4\mu}\int_0^R r(r^2-R^2)dr,$$

Note that we need to remember our factors of r and π when integrating over the surface of the pipe inlet in cylindrical coordinates. Performing the integral we obtain

$$Q=\frac{\pi\Delta P R^4}{8\mu L},$$

or written as the common pressure-flow relationship,

$$\Delta P=Q\frac{128\mu L}{\pi D^4}$$

Notice the term $128\mu L/\pi D^4$ is the hydraulic resistance which relates pressure and flow. The pipe resistance is a strong function of the diameter. One effect is the area, there is less volumetric flow if the area

is reduced (for the same velocity). The other effect is that viscosity is stronger when the diameter is small.

Taking the relationship for pressure and flow in a pipe and recasting in terms of the velocity, we could rearrange the equation to obtain,

$$\Delta P = 32\frac{\overline{u}\mu L}{D^2}.$$

Here $\overline{u}$ would be the average velocity in the pipe, i.e. $Q/(\pi D^2/4)$. Remember from an earlier chapter that pressure drop is pipe flow is characterized as

$$\Delta P = \frac{1}{2}\rho\overline{u}^2\frac{L}{D}f(\mathrm{Re}).$$

Equating the last two expression yield,

$$32\frac{\overline{u}\mu L}{D^2} = \frac{1}{2}\rho\overline{u}^2\frac{L}{D}f(\mathrm{Re}) \quad \rightarrow \quad f = \frac{64\mu}{\rho\overline{u}D} = \frac{64}{\mathrm{Re}}.$$

the analytical solution for laminar flow friction factor that we saw on the Moody diagram in Chapter 3. As we have described before, this solution is only observed up to around a Reynolds number of 2300 and then stability is lost and flow become turbulent.

11.9 Comments on the stability of solutions

In simple fluid flows as we have described in this chapter, we can often find a solution to the Navier Stokes equations. As we have discussed, these solutions are not guaranteed to be stable. Imagine trying to balance a pencil on its sharpened tip. While a free body diagram of a vertical pencil on a table is a perfectly valid theoretical solution, you can easily see that the situation is highly unstable and not realizable in practice. If the pencil leans ever so slightly to one direction, the pencil falls over. Now imagine an unsharpened pencil. You can balance this one on its end, but if you hit the table it falls over. Both of these situations can occur in fluid flows where the solution to the equations is valid, however it is not stable to either finite or infinitesimal perturbations. There are many routes to instability in fluid flows, many of which are an active area of research. Since the Navier-Stokes equations are non-linear there are no guarantees of *unique* solutions to problems.

However it is the issue of stability and non-linearity that make fluid mechanics a fascinating area of study.

The most straightforward method of stability analysis is to take a situation where a simple solution to the Navier-Stokes equations is known. We can then consider very small perturbations to this solution and then decide if those perturbations grow or decay in time. Mathematically we make a substitution where we would say the the velocity is $u = u_0 + \hat{u}$ where u_0 is the known simple base state and $\hat{u}$ is an infinitesimal perturbation to that base state. We can do this for all the variables and plug this into the Navier-Stokes equations. Since the perturbations to the base state are assumed small, we can discard all the non-linear terms (products of hat terms) which comprise the product of two small things. The result are linearized partial differential equations for the perturbations to the base state. In practice, we can usually solve this linear problem (at least numerically). We can then look at the solutions and see if they either grow or decay with time. If the solutions to the perturbed equation grow with time, this indicates that if we provide a slight random kick, the solution will grow in size until it dominates the problem, noting that the flow is unstable. This type of analysis is called linear stability analysis.

This type of stability prediction works remarkably well in some cases and remarkably bad in others. Two cases where this analysis works well are called Rayleigh-Benard convection and Taylor-Couette flow. Rayleigh Benard convection is what occurs when a channel is heated from below and the hot fluid near the lower surface wants to rise up due to buoyancy. If the temperature difference is small the fluid stays still and heat is transported by conduction. As the temperature difference increases, convection cells form. These convection cells can result in interesting large scale patterns in the flow, such as stripes and spirals. The initial onset of convection is well-predicted by a linear stability analysis.

The Taylor-Couette problem is the flow between two concentric cylinders rotating relative to each other. If the inner cylinder is rotating slowly, we have a Couette flow in the gap between the cylinders that we studied earlier. At higher speeds there is a centrifugal instability that causes a set of convection cells to set in. So rather than a simple shearing motion, a set of vortices spontaneously emerge that wrap around the cylinder. An example image is shown in Figure 11.8. If one

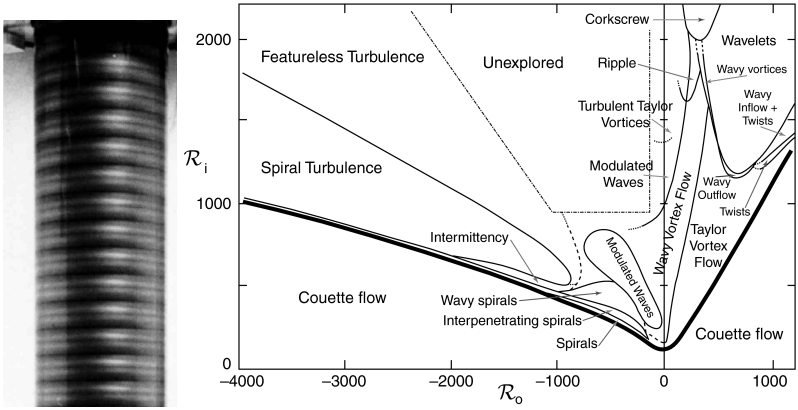


Figure 11.8 Image of the unstable vortices that wrap around the cylinder in a Taylor-Couette device. On the right, the phase diagram of the different types of flow observed if one can independently rotate the inner and out cylinders in the device. Image from Album of Fluid Motion. Phase diagram from Anderdeck, Liu, and Swinney, J. Fluid Mech 1986.

controls the speed of both the inner and outer cylinder the flow is extremely complex and many different regimes of qualitative behavior are seen in the phase diagram of Figure 11.8. The lower boundary of the phase diagram from simple sheared Couette flow to the initial instability is well predicted by linear stability analysis. The threshold and flow patterns that one observes in experiment closely match those that are predicted by linear analysis. The rest of the phase diagram beyond this initial instability is much more complicated to predict. As you exceed the threshold non-linear effects come into play and the patterns become ever more complicated and beautiful. While the topic of linear stability analysis is beyond what we will do in this class, it is not that difficult. It is a topic found in a number of textbooks.

Flow in a pipe is a case where the linear stability analysis works remarkably bad. In this case, the analysis predicts the flow is stable at all Reynolds numbers, though we know in practice that is not true. The mechanism for transition to turbulence in simple pipe flow is a problem which is still worked on extensively to this day. While a lot is understood about the laminar to turbulent transition in this problem,

it is still an area of research 125 years after Reynolds first made his observations.

There are many complications with the Navier-Stokes equations but the point I want to emphasize is simple. It is not sufficient to simply solve the Navier Stokes equations. We always need to ask whether those solutions are stable. Predicting stability is sometimes straightforward and sometimes not. Just be aware that asking the question of stability is a big deal in fluid mechanics and the complications of stability lead a number of interesting and beautiful phenomenon. In the end, however, any analysis or simulations you conduct must be brought before the ultimate judge - experiments.

11.10 Computational Fluid Dynamics (CFD)

Since the 1970s, the desire to simulate fluid flow has been one of the great drivers for development of simulation software, numerical methods, and supercomputers. CFD remains a very large endeavor in both industrial and academic settings. Due to Moore's law and dramatic increase in computing power, the possibilities with CFD changes quite rapidly. Problems that would have challenged the fastest computers and the best researchers some years ago can be done routinely with commercial software and a typical personal computer. Based on looking at the sample problems that come with modern CFD packages, I would estimate that a problem that had the complexity to have been a PhD thesis 10-15 years ago can be done routinely with commercial software. In terms of practical available computing power I would also estimate about a 10 year lag between when a cutting edge supercomputer problem becomes feasible with a typical low-cost computer. These two forces, improving hardware and software, continually lower the financial cost and required expertise to using CFD.

There are different numerical methods which all approximate the solution of the Navier Stokes equations as discrete chunks. The most common for commercial CFD packages are based on either finite element, finite volume, or finite difference methods. I suggest that if you ever work with CFD for a job or in graduate school you learn the difference. For linear problems such as heat conduction, a commercial package can accurately solve the heat equation for any arbitrary 3D

geometry with very little effort. A novice user can quickly generate a good simulation.

Due to the non-linear nature of the Navier-Stokes, we are not yet at a state of "plug and play" for all problems. Laminar single phase flows are at that point today. If we have a 2D or 3D flow where the Reynolds number is low enough that the non-linear terms are not too "strong", then a modern CFD package can usually do a good job without the user getting too involved in the details. This does not mean that laminar flow is always easy, that there aren't sometimes difficulties, that you should run to the computer and shy away from analysis or that you are always going to get a physical answer. But practically speaking, if you are interested in a single phase laminar flow you could probably simulate everything you need without too much trouble or expertise.

All the examples we worked in this section could be set up and run very quickly using a commercial CFD package. However, as we discussed earlier in the heat conduction chapters, when you just do a simulation you can miss simple scaling laws, and simple formulas that are good for basic understanding of trends and also very useful in design problems. Simulation is another tool that can work side by side with good analysis, but is rarely the substitute.

We will discuss turbulent flows in a later chapter, but simulating turbulent flows is a much trickier situation and one where you should know what you are doing. There are fundamental limitations that make high Reynold's number flows challenging that we will address later. For now, it is worth noting that if you use a CFD package and get good results, turn up the Reynold's number and it is guaranteed to stop working.

12

Inviscid flow, Euler's equation and Bernoulli

Previously we derived the Navier Stokes equations for a Newtonian fluid with viscosity. If we take the limit of the Navier Stokes equations of zero viscosity, we recover the Euler equations,

$$\nabla \cdot \mathbf{v} = 0, \tag{12.1}$$

$$\rho \frac{D\mathbf{v}}{Dt} = -\nabla P + \rho \mathbf{g}. \tag{12.2}$$

We previously found that for a human swimming at a reasonable pace, the Reynolds number was on the order of a million. We can imagine for a car going down the highway it is even larger. For weather related phenomena a Reynolds number of 10^{10} to 10^{15} would not be uncommon. Since the Reynolds number is a measure of the ratio of inertia to viscosity, it seems that perhaps dropping the viscous terms and turning to Euler's equation might be a good approximation for high Reynolds numbers. Euler's equation looks simpler but it's behavior can be quite different than the Navier-Stokes even in the limit of high Reynolds number. The mathematical difference is in the number of boundary conditions required. For flow over a solid surface, Euler's equation would only require (and could only enforce) the boundary condition that flow does not penetrate the surface. Fluid would slip over the surface in Euler's equation. Since the Navier Stokes equations has higher spatial derivatives (in the viscous terms), it requires more boundary conditions. The Navier Stokes requires that flow does not penetrate the surface *and* that the flow does not slip at the surface. This subtle difference can lead to quite different behavior. Mathematically, the zero viscosity limit is called a singular perturbation. This

means that we take the limit of the small parameter (viscosity) going to zero and we fundamentally change the problem at hand.

In the last chapter on Navier-Stokes solutions we studied examples where the balance of forces was between viscosity and some driving force such as applied pressure. In all the problems in the last chapter, the fluid had no acceleration. When using Euler's equation we are ignoring viscosity and the balance of forces is between acceleration and pressure (or gravity). A good strategy for cases of high Reynolds number flows where fluid acceleration is important is to start a problem by assuming that Euler's equations work. We could then check what the predictions and trends are versus what experiments and experience say to decide if the approximation was reasonable. We will begin to understand the breakdown of Euler's equations better after the next chapter when we discuss boundary layers.

12.1 Flow along a streamline: Bernoulli

One of the most useful (and simple) relations for flows with no viscosity is known as Bernoulli's equation. Let's start by a simple demonstration of the where the derivation comes from and then we will follow with a more general derivation.

Consider the flow through a constricted tube as shown in Figure 12.1. Let's consider steady, incompressible, and inviscid flow. For now, let's further neglect gravity. Without gravity, Euler equations become

$$\rho \mathbf{v} \cdot \nabla \mathbf{v} = -\nabla P.$$

The balance here is that acceleration of the fluid particle is balanced by the net force due to the pressure gradient. In component form the x momentum equation is

$$\rho \left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = - \frac{\partial P}{\partial x}.$$

Now consider the flow just along the centerline. By symmetry, we could argue that there is no component of velocity in the y or z direction; $v = 0$ and $w = 0$. Along the centerline, there is nothing interesting contained in the y and z momentum equations since all accelerations in those directions are zero (you can write out all the components and

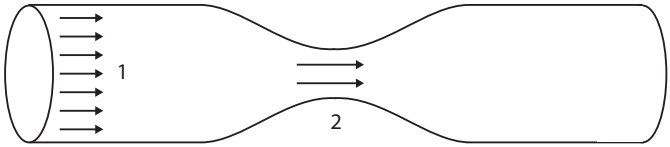


Figure 12.1 Venturi device. The cross sectional area at point 2 is less than at point 1, thus the flow accelerates. Since the velocity at the throat is faster than the upstream, Bernoulli's equation predicts the pressure is lower at the throat.

convince yourself this is true). Along the centerline the x momentum equation simplifies to,

$$\rho u \frac{\partial u}{\partial x} = -\frac{\partial P}{\partial x}.$$

Using the chain rule note that,

$$\frac{\partial\, u^2}{\partial x} = u\frac{\partial u}{\partial x} + u\frac{\partial u}{\partial x} = 2u\frac{\partial u}{\partial x}.$$

Using this result we can rewrite the momentum equation as

$$\frac{\partial \frac{1}{2}\rho u^2}{\partial x} = -\frac{\partial P}{\partial x},$$

or

$$\frac{\partial}{\partial x}\left(\frac{1}{2}\rho u^2 + P\right) = 0.$$

Along the centerline,

$$\frac{1}{2}\rho u^2 + P = \text{Constant},$$

which we will soon see is a specific version of a more general relationship called Bernoulli's equation.

For the device shown, by conservation of mass we know that the flow velocity will increase at the throat. Our expression for Euler's equation along the centerline says that if the velocity increases, the pressure must decrease. When the flow accelerates, the pressure decreases. When the flow decelerates, the pressure increases. This general trend should

make sense from the perspective of the fluid particle. If you are flowing through the device and are accelerating upon going into the throat, then there has to be some net force acting on you pushing you from behind. This net force is the high pressure behind you and lower pressure in front of you.

12.2 General Bernoulli

Let's consider a more general derivation of Bernoulli's equation. Again, we will consider steady incompressible, constant density, inviscid flow where the conservation of momentum would be written as

$$\rho \mathbf{v} \cdot \nabla \mathbf{v} = -\nabla(P + \rho g z).$$

Here we assumed that gravity is pointing in the negative z direction, without loss of generality. If we look up a table of vector calculus identities we are free to write the term of the left side in a number of different forms. One identity would allow us to write,

$$\rho \left(\frac{1}{2} \nabla \mathbf{v}^2 - \mathbf{v} \times (\nabla \times \mathbf{v}) \right) = -\nabla(P + \rho g z).$$

which, when density is constant, is equivalent to

$$-\rho \mathbf{v} \times (\nabla \times \mathbf{v}) = -\nabla \left(\frac{1}{2} \rho \mathbf{v}^2 + P + \rho g z \right).$$

We discussed this previously, the curl of the velocity vector $\nabla \times \mathbf{v}$ is also known as the vorticity ω. Vorticity is a measure of the local solid body rotation of a fluid particle. It can be useful in a number of settings, but it is a little hard to wrap your head around what it means. Basically if you measured a flow and plotted an image of the vorticity field, it would be highest where the fluid was spinning (i.e. in tornadoes, toilets, and whirlpools). It is a property of the curl operator, *the vorticity always points perpendicular to the velocity vector*. If you have a 2D flow in the plane of a sheet of paper, the vorticity will point out of the page. We will discuss vorticity more in a coming section, for now you can consider it a mathematical thing. Rewriting conservation of momentum with the vorticity yields,

$$-\rho \mathbf{v} \times \omega = -\nabla \left(\frac{1}{2} \rho \mathbf{v}^2 + P + \rho g z \right).$$

Note that at this point we really haven't done anything, just rewritten the same equation in a different form using some vector calculus identities.

Let's take the whole equation and take the dot product with the velocity vector itself,

$$-\mathbf{v} \cdot \left(\rho \mathbf{v} \times \omega = -\nabla \left(\frac{1}{2} \rho \mathbf{v}^2 + P + \rho g z \right) \right).$$

Notice the term on the left side. The curl of the velocity and the vorticity vectors will always be perpendicular to those two vector. Thus the dot product with the velocity vector itself will be zero. This fact can also be easily proven by carrying out all the terms in component form. Since the left side is always zero, we obtain

$$0 = \mathbf{v} \cdot \nabla \left(\frac{1}{2} \rho \mathbf{v}^2 + P + \rho g z \right).$$

This fact means that the quantity in parenthesis *does not change in the direction of the velocity vector*. In a fluid flow we define a streamline as a line that follows the velocity vectors. In a steady flow a streamline will correspond to the path a blob of injected dye would follow. Thus in a steady flow,

$$\frac{1}{2} \rho \mathbf{v}^2 + P + \rho g z = \text{a constant along a streamline.} \tag{12.3}$$

This equation is known as Bernoulli's equation. It is based on some restricted assumptions, namely 1) incompressible, 2) steady, and 3) inviscid flow. Despite these restrictions it is a powerful equation because of its simplicity.

Bernoulli's equation has a simple interpretation. The kinetic energy per unit volume is $\frac{1}{2} \rho \mathbf{v}^2$, and the potential energy is $\rho g z$. These expressions should look familiar from particle mechanics only here we use the mass density rather than the total mass. Pressure exerts a force per unit area, thus a change in pressure between two locations indicates that the pressure is doing work on the fluid. This work is related to the familiar expression that work is equivalent to the integral of force over distance. Bernoulli's equation is saying that the work done by pressure is equal to the change in energy (kinetic plus potential).

While simple and powerful, Bernoulli's equation can also be misleading. There is a tendency to want to use it under conditions that are

not appropriate. We must always ask if the assumptions are met when using Bernoulli, and ultimately determine whether this simple equation provides predictions which match reality. The most common mistake is using Bernoulli's equation when viscosity is not negligible.

12.2.1 Example: Venturi

A venturi meter is a simple device for measuring flow rate shown in Figure 12.1. The meter is inserted in a pipe and you measure the pressure difference between an upstream location and at the center of the contraction. Gravity acts normal to the flow, thus the potential energy does not change for a fluid particle going through the meter. If the area change is known, the velocity ratio at points 1 and 2 is known as

$$\frac{v_2}{v_1} = \frac{A_1}{A_2}$$

from conservation of mass. Here v_1 and v_2 are average velocities. Applying Bernoulli from point 1 to 2 yields

$$\left(\frac{1}{2}\rho v^2 + P\right)_1 = \left(\frac{1}{2}\rho v^2 + P\right)_2 .$$

Combining these expression yields,

$$P_1 - P_2 = \frac{1}{2}\rho {v_2}^2 - \frac{1}{2}\rho {v_1}^2 = \frac{1}{2}\rho v_1^2 \left(\left(\frac{A_1}{A_2}\right)^2 - 1\right)$$

If the pressure difference $P_1 - P_2$ is measured and the area ratio is known, then we can figure out the fluid velocity. The venturi is one of the simplest flow meters.

An old use of a venturi is the carburetor. The carburetor is a device for mixing fuel and air prior to combustion in an engine. They are not used in cars anymore, replaced by fuel injectors, however they are still in use in low-cost engines. The carburetor has a venturi and makes use of the low pressure in this region to draw fuel into the airstream.

12.3 Euler equations across a streamline

We can also use Euler's equation to provide a simple relationship for what happens across streamlines. The following derivation can be made

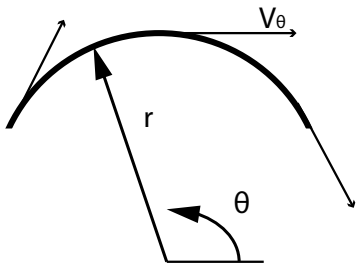


Figure 12.2 Flow around a circular streamline. Euler's equations can tell us how the pressure changes as we jump across a streamline.

more general, but for simplicity lets consider that the streamlines at some point in the flow are curved as perfect circles, such that we can describe the flow in cylindrical coordinates. The nice thing about our vector calculus approach is that we simply look up the different operators in different coordinate systems, and we are good to go. Looking up the operators for cylindrical coordinates where the flow is in the $r-\theta$ plane and gravity only points in the z direction the Euler equations in steady flow become,

$$\rho\left(u_r\frac{\partial u_r}{\partial r}+\frac{v_\theta}{r}\frac{\partial u_r}{\partial \theta}-\frac{v_\theta^2}{r}\right)=-\frac{\partial P}{\partial r}$$

$$\rho\left(u_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}\right)=-\frac{1}{r}\frac{\partial P}{\partial \theta}$$

We use the subscripts θ and r just as reminders that u and v are the components in these coordinate directions. Since at this point, the streamlines follow the circle, $u_r = 0$. There is no radial flow. Likewise, since we are following a circle here $\partial u_r/\partial\theta = 0$. Following the circular streamline at this point allows us to remove some terms from the equation to obtain equations that are valid on the streamline,

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

$$\rho\left(\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}\right)=-\frac{1}{r}\frac{\partial P}{\partial \theta}.$$

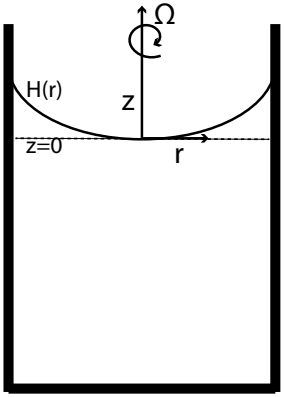


Figure 12.3 Schematic for the example of computing the free surface shape of a cylindrical tank of water in solid body rotation.

The second equation for the θ component of momentum is equivalent to

$$\frac{\partial}{\partial \theta}\left(\frac{1}{2}\rho v_\theta^2 + P\right) = 0$$

which states that Bernoulli holds in the θ direction. Since this direction is the streamline, we are satisfied.

The new equation is that the pressure changes *across* the streamlines as

$$\rho\frac{v_\theta^2}{r} = \frac{\partial P}{\partial r} \quad \text{across a streamline} \tag{12.4}$$

Equation 12.4 should make some physical sense. If we have flow spinning a circle, the pressure increases as we move outward in radius. The pressure gradient is equivalent to the tension of a string that holds a rock that you swing in a circle. If the streamlines are straight, then there is no pressure jump across them.

12.3.1 Example: rotating tank

Consider a container of fluid on a turn table such that the whole system is in solid body rotation. The velocity field of the fluid is $v_\theta = \Omega r$ where Ω is the constant angular velocity and r is the coordinate. The axis of

rotation and gravity are aligned such that gravity does not appear in the radial momentum equation. Using our expression from the previous section, the radial component of the momentum equation is

$$\rho \frac{v_\theta^2}{r} = \frac{\partial P}{\partial r} \quad \rightarrow \quad \rho \Omega^2 r = \frac{\partial P}{\partial r}.$$

Since everything on the left side is a constant we can integrate this equation with respect to r (holding $z = Z$ constant) from the origin to obtain the pressure distribution as a function of r.

$$P(r, Z) - P(r = 0, Z) = \frac{1}{2} \rho \Omega^2 r^2$$

The pressure is higher at the wall of the container than the center. The vertical momentum equation is simple since there is no vertical flow, namely, at a constant radial location ($r = R$),

$$P(z, R) - P(z = 0, R) = -\rho g z.$$

If we set $z = 0$ to the surface of the liquid at the center ($r = 0$) of the container, we can set the pressure reference at this point to be 0. We can follow at radial line outward to the wall to an arbitrary point r,

$$P(r, z = 0) = \frac{1}{2} \rho \Omega^2 r^2.$$

Vertical balance provides the pressure at the same point to be

$$P(r, z = 0) = \rho g H(r) - P(z = H, r)$$

Since the pressure at the surface of the liquid is zero, $P(z = H(r), r) = 0$, then we are left with a simple result that the height of the fluid surface is,

$$H(r) = \frac{1}{2} \frac{\Omega^2 r^2}{g}.$$

You can confirm the result that the surface of the fluid is a parabola by spinning a cup of fluid. You will see the shape of the surface looks parabolic. We could have obtained this result using dimensional analysis quite readily. We would not have known the factor of 1/2 if we had done dimensional analysis.

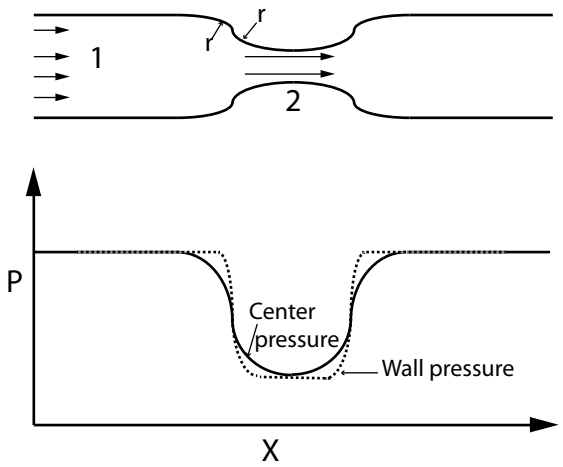


Figure 12.4 Schematic of the pressure distribution in a venturi meter comparing the center pressure to the wall pressure.

12.3.2 Example: Venturi revisted

Let's now reconsider the pressure distribution in a venturi meter. The pressure along the centerline will follow that predicted by the Bernoulli equation along the streamline. The pressure at the wall will be different. In the first part of the contraction, the radius of curvature of the streamline following the wall is such that the pressure at the wall would be *h*igher than the pressure along the center. Approaching the nozzle throat, the wall streamline passes through an inflection point and the radius of curvature changes sign. On the final approach to the throat, the pressure along the wall will be *l*ower than the pressure along the centerline. The pressure profile as a function of distance is shown schematically in the dotted line of Figure 12.4. While a more detailed calculation would be required to quantitatively describe the difference, the qualitative result that pressure must increase as we move outward along a radius of curvature is sufficient to explain the basic effect we would see in experiment.

12.4 Vorticity

This section is going to rely on a fair amount of vector calculus and identities. If you do not remember all these identities, then don't get too worried. Try to follow the derivation, but focus on understanding the consequences. First let's restate that vorticity is a vector defined as the curl of the velocity,

$$\omega = \nabla \times \mathbf{v}. \tag{12.5}$$

Vorticity is a measure of solid body rotation of a fluid particle. Note that flow can go in a circle, and still have no vorticity. A Ferris Wheel is an example of a circular flow where the fluid particle (i.e. you) does not undergo solid body rotation. You face the same direction as the wheel spins. On a Merry-Go-Round, you the fluid particle, certainly are rotating which is why you get very dizzy. In a fluid, water waves is an example where fluid traces a circular path as a wave passes over, however the fluid particles do not undergo solid body rotation (i.e. the vorticity is zero).

You can construct a simple vorticity meter. You only need a little paddlewheel that is free to spin on an axle. If you stick a paddlewheel in a flow and it spins, then there is a component of vorticity pointing along the wheel's axis of rotation. If you move the paddlewheel's orientation around until it spins the fastest, then that would be the axis along which the vorticity vector points.

The total vorticity over an area can be related to the velocity field using another vector calculus theorem, Stokes's Theorem. This theorem says,

$$\Gamma = \oint \mathbf{v} \cdot \mathbf{t} dl = \int (\nabla \times \mathbf{v}) \cdot \mathbf{n} dS. \tag{12.6}$$

This theorem says that if I integrate the tangential velocity ($\mathbf{t}$ is the tangent vector of the curve) around some arbitrary closed loop, where the line element length is dl, this quantity must equal the net vorticity flux coming through the area. The value of the integral is defined as a property, the circulation Γ. The circulation tells you if the fluid is, well, circulating. Stoke's theorem is a general vector calculus theorem for any vector, but here I have written it in the language of velocity and vorticity.

You should look up a weather map for a Northern Hemisphere At-

lantic hurricane which has wind vectors drawn on it. You will notice the vectors show the winds going in a circle in the counterclockwise direction. To estimate the circulation, draw an arbitrary circle which roughly follows the winds. To estimate the magnitude of the circulation simply estimate the average wind speed on your circle, and multiply that speed by the circumference. To compute the circulation more accurately you could break your circle into little pieces to account for the fact that the wind speeds will vary around the circle. Adding up these little pieces is the approximation of the integral in the formal definition.

Now let us derive an equation for the vorticity. Lets take the curl of Euler's momentum equation in constant density, incompressible flow,

$$\rho\nabla\times\left[\left(\frac{\partial\mathbf{v}}{\partial t}+\mathbf{v}\cdot\nabla\mathbf{v}\right)=-\nabla(P+\rho gz)\right]. \tag{12.7}$$

Now we need to recall our vector calculus identities. You might remember that the curl of the gradient of something is zero. If you don't believe this, you could carry out the operation for each component to find out. There are also identities that allow us to expand the "v dot grad v term".

$$\frac{\partial\omega}{\partial t}+\mathbf{v}\cdot\nabla\omega-\omega\cdot\nabla\mathbf{v}=0 \tag{12.8}$$

which is the same as

$$\frac{D\omega}{Dt}=\omega\cdot\nabla\mathbf{v}. \tag{12.9}$$

What is interesting is that the pressure and gravity have disappeared. Vorticity is a measure of solid body rotation and pressure and gravity act through the fluid particles center of mass in constant density flows, thus they can not change the rotation. If there are density gradients in the fluid, then these can couple with gravity to create vorticity. This mechanism of vorticity production is common in oceanic and atmospheric flows. Now lets turn to trying to interpret these equations starting with a simple example of 2D flow, remembering the restriction that we have assume inviscid flow.

12.4.1 2D flows

Lets restrict our analysis to 2D flows to make things a little easier. First of all in a 2D flow, velocity only has components in u and v and

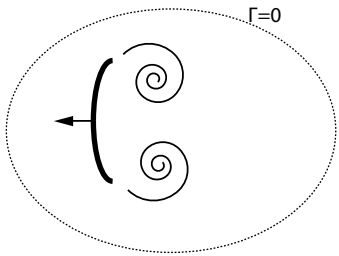


Figure 12.5 Drag a spoon slowly through a bowl of water. You will see two vortices form behind the behind of opposite sign but equal strength. The two vortices will cancel each other such that the total circulation around the outer dashed contour is zero.

gradients in x and y. Vorticity in a 2D flow only has a component in the z direction; $\mathbf{v} = [u\ v\ 0]$; $\omega = [0\ 0\ \omega_z]$. Thus the term in Equation 12.9 on the right hand side is zero since the vectors are not in the same plane. In a 2D flow,

$$\frac{D\omega}{Dt} = 0 \quad \text{in 2D flows.}$$

This equation states that vorticity is a material property. Whatever vorticity the fluid particle has initially, it retains that amount and then simply goes with the flow. Since the vorticity for a fluid particle is conserved, if we draw a loop which is a material loop (a massless, strechable string with no elasticity that can go with the flow), the vorticity inside that loop is the same for all time. Thus the integral of the vorticity or the circulation is constant,

$$\frac{D\Gamma}{Dt} = 0 \quad \text{around a closed material loop.}$$

This statement is known as Kelvin's circulation theorem. It is an interesting theorem and is more a general theorem than the simple argument I made here. A formal proof of the theorem would show that the theorem is valid if Euler's equations hold *along the contour around which the circulation is calculated.* Thus, there can be viscosity inside the closed loop. The theorem also holds in three dimensions.

So how does this theorem manifest itself? Try the following experiment. Take a bowl of water and sprinkle some tracer particles in the water; fine black pepper works well. Take a spoon and give the sur-

face of the fluid a little flick. You should see two vortices generated. They will be of opposite spin and propagate themselves for a bit before decaying away. Now, imagine you draw your loop to calculate the circulation around the spoon and far away from it. Since everything is at rest intially, the circulation is zero. Now you flick the spoon and create your vortices. So even though viscosity has done something inside the loop (we could not create the vortices without it), the circulation theorem still applies far away from the loop. So the circulation is still zero even though there is vorticity inside the loop. The circulation theorem states that the circulation is zero and remains so. This means that the positive and negative vorticity you generated must cancel each other out via the definition of circulation as the vorticity integrated over the area (via Stokes's theorem). In this experiment you get two vortices of equal strength and equal size, such that the total integrated vorticity is zero. The theorem tells you nothing of the vortices decay, just that the total circulation should always be zero.

You may have noticed that the equal sized but oppositely signed vortices propagate themselves in a straight line. This behavior can be understood by using an ideal vortex which is a solution to Euler's equation. The ideal vortex has a velocity field given as

$$v_\theta(r) = \frac{\Gamma}{2\pi r}$$

This velocity field is provided in cylindrical coordinates, thus the flow is in the θ direction but only depends on r. This velocity field can be plugged into Euler's equation and would be found to be an acceptable solution. It is easy to show that the circulation of this vortex is

$$\int_0^{2\pi} v_\theta(r) r d\theta = \int_0^{2\pi} \frac{\Gamma}{2\pi r} r d\theta = \int_0^{2\pi} \frac{\Gamma}{2\pi} d\theta = \Gamma.$$

The circulation is a constant regardless of the radius of the circle we choose to perform our calculation along. The vorticity, which only has a component in the z direction is,

$$\omega_z = \frac{1}{r}\frac{\partial r v_\theta}{\partial r} = \frac{1}{r}\frac{\partial}{\partial r}\frac{\Gamma}{2\pi} = 0$$

zero everywhere, except at the origin where it is undefined.

Now consider two vortices of equal circulation but opposite sign, separated by some distance, Figure 12.6. One vortex will induce a velocity

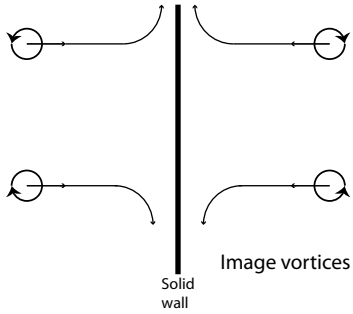


Figure 12.6 Motion of two ideal vortices of opposite sign approaching a wall. The velocity field of each vortex propagates the other. As the vortices approach the wall, the introduction of an imagined image vortex can meet the boundary condition at the wall of no flow penetrating the surface. As the four vortices approach the wall, the induced velocity fields cause then to spread apart.

field that will push the other forward. Since the vortices just move with the fluid, that is what $D\omega/Dt = 0$ says, then you can think about the velocity of one vortex simply acting to move the other. Since the vortices are the same strength they push each other in a straight line. If you created two vortices of equal strength but the same direction of the spin, the two vortices would orbit each other in a circle.

Now repeat your experiment with the spoon near the wall of your bowl. You should be able to see that the vortices propagate themselves to the wall, and then spread apart as they approach the wall. This can be understood by imagining a set of image vortices on the other side of the wall. These image vortices don't really exist, but you should be able to convince yourself with symmetry arguments that the flow generated by the image vortices will always cancel the real vortices such that there is no velocity penetrating the wall. The flow on the left side of Figure 12.6 where the real vortices exist is the same whether there was the wall or the set of symmetric image vortices. Figuring out the qualitative flow is easier by visualizing the image vortices. Now as the four vortices approach the wall, you can see that the image vortices push each other outward if you consider the action of each vortex on the other.

12.4.2 3D flows

In 3D, the above analysis just gets a little more complicated. First of all, the circulation theorem remains, so without proof we will simply state that in 3D the circulation is constant around a material loop. However, simple applications of the circulation theorem (at least in this course) are usually simple 2D approximations to give us some qualitative understanding of the flow.

The vorticity equation in 3D is,

$$\frac{D\omega}{Dt} = \omega \cdot \nabla \mathbf{v}.$$

and has a non-zero term on the right hand side. Vorticity for a material point is not constant in 3D as there is an extra term in the equation. This extra term accounts for things which cannot occur in a 2D flow. One effect is vortex stretching. If a vortex is stretched out by a flow, it intensifies, just like an ice skater speeding up as they pull their arms in. If a vortex is squashed, the vorticity decreases. The stretching of the vortex can be see in any bathtub drain. The vortex gets intensified as it stretches down the drain. Another easy experiment to see vortex stretching is to take a soda bottle and fill it with water. Turn it upside down over the sink and as it is glugging, give it a strong swirl by hand. You will set up a vortex that will be intensified as it is stretched and pulled down out of the bottle. There are also effects that occur due to vortex tilting. Tilting a vortex line can change a particles rotation. This effect would be analogous to the physics demo where the instructor takes a spinning bicycle wheel and tilts it while sitting on a stool which is free to rotate.

In vortex dynamics we refer to vortex lines and tubes. A vortex line is one which follows the vorticity vector, like a stream line. It would be the axis of a tornado. A vortex tube would be a collection of vortex lines and the vorticity vector is everywhere parallel to the surface of the vortex tube. In 3D there are three laws of vortex motion that were derived by Helmholtz in 1858. These laws are good for the approximations we have been dealing with - inviscid and constant density. Helmholtz's laws are (Saffman (1992)).

- Fluid particles initially free of vorticity remain free of vorticity.
- Vortex lines and tubes move with the fluid.

- The strength of a vortex tube does not vary with time.

Due to the definition of vorticity as the curl of the velocity, it true that $\nabla \cdot \omega = 0$ since the divergence of the curl of any vector is zero (you should quickly see if you can prove this fact to yourself). Since vorticity is divergence free, then a vortex tube must have constant strength along its length. While this last statement might not sound obvious, the proof is precisely the same as showing that the total volumetric flow rate through a pipe of varying cross section area must be the same at every cross section if the flow is incompressible. If a flow is incompressible, $\nabla \cdot \mathbf{v} = 0$. The walls of the pipe are like the walls of the vortex tube - since by definition there is no vorticity normal to the surface of the vortex tube. At every cross section of the tube the total flux of vorticity, $\int \omega \cdot \mathbf{n}$, is a constant just like in a pipe $\int \mathbf{v} \cdot \mathbf{n}$ is a constant. If the strength of the vortex tube is constant everywhere along the length, then vortex tubes must either close on themsleves (i.e. a smoke ring), go to infinity, or end on solid boundaries. An example of vortex tubes are the long white contrails seen behind airplanes on clear days when the weather conditions are right. The two long white streaks are intense vortex tubes which extend far across the sky. If you observe these streaks closely you will notice that some distance away from the plane, the individual tubes start developing a wavy character. These waves are due to a fluid instability known as the Crow instability. Notice that as the amplitude of the waviness increases the two tubes will interact, cross, reconnect, and form a series of vortex loops. These circles are now closed vortex tubes. How well you can make these observations depends on the weather as the level of atmospheric turbulence can disrupt the vortex visualization.

12.5 Irrotational flow

In a 2D flow we said that the vorticity equation says that $D\omega/Dt = 0$, the vorticity remains constant from a material point of view. Thus, if a flow starts with no vorticity, none can be created. This statement was Helmholtz's first vortex law. You need viscosity to create and destroy vorticity, thus you can't make vorticity with Euler's equations. You can also see by the general 3D version of the vorticity equation, that if the

flow has zero vorticity initially, none can be created. Therefore, it is common when dealing with Euler's equations to consider irrotational flow, that is a flow where $\omega = 0$ everywhere.

If the flow has no vorticity, then you might remember a trick used in electrostatics. The curl of the gradient of a scalar function is always zero, this statement is an identity from vector calculus. Therefore, if the flow has no vorticity (curl), then we can define a velocity potential,

$$\mathbf{v} = \nabla\phi,$$

which is 100% analogous to the electric potential used in electrostatics. If we can define a velocity potential, then conservation of mass $\nabla\cdot\mathbf{v} = 0$ yields

$$\nabla^2\phi = 0.$$

This equation is known as Laplace's equation. At a solid surface we would have the boundary condition that $\nabla\phi\cdot\mathbf{n} = 0$ which says that the flow cannot penetrate the surface; $\mathbf{v}\cdot\mathbf{n} = 0$. This equation is really easy to solve, especially numerically if you have a complicated geometry. Once you have the velocity you can easily calculate the pressure, especially in a steady flow. If you go back to our derivation of Bernoulli, you will find that we wrote the momentum equation in the following form,

$$\rho\mathbf{v}\times\omega = -\nabla\left(\frac{1}{2}\rho\mathbf{v}^2 + P + \rho gz\right).$$

If the vorticity is zero, then it is clear that the left hand side is zero. Therefore, in an irrotational flow Bernoulli's equation holds *everywhere*, not just along a streamline. Therefore,

$$\frac{1}{2}\rho\mathbf{v}^2 + P + \rho gz = \text{constant everywhere for irrotational flow.}$$

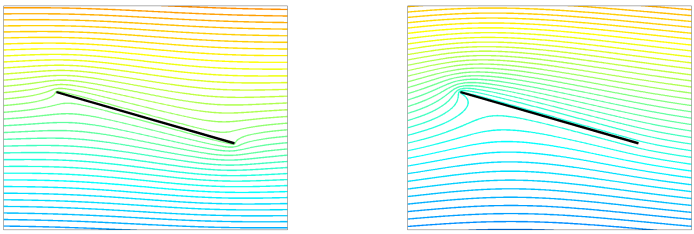
So in irrotational flow, you solve Laplace's and then plug the velocity field into Bernoulli's equation to solve for the pressure. This is a very simple theory which is quite amenable to solution, which is in contrast to the money you would earn for a general solution to the Navier-Stokes equations. However, the utility of irrotational flow is quite limited. There are a number of good books that are filled with solutions to potential flow and a number of good online calculators that will solve

potential flow problems for you. Potential flow finds a few niche applications, can many times (especially in aerospace applications) provide a good qualitative picture of the flow and the theory is extremely important in the history of the development of the field of fluid dynamics. However, the practical utility of potential flow is limited.

12.6 Lift on an airfoil

Euler's equations can be used to predict lift forces on an airfoil. Euler's equation, since it has no viscosity, cannot predict drag. Imagine an airfoil with flow at uniform velocity coming at it. We can use our equations for irrotational flow to solve this problem. We can assume the flow is irrotational because imagine the airfoil is at rest. There is no motion and thus no vorticity. Let's accelerate our airfoil, and since this is an Euler flow, we create no vorticity. The flow is irrotational and we can then solve $\nabla^2\phi = 0$ around the airfoil shape to obtain the steady flow patterns. The solution would show that streamlines approach and leave the airfoil at two points near the leading and trailing edge. However the flow does not match what we observe experimentally. The flow experimentally, shows that the streamlines leave the body at the trailing edge. The only way we can make the calculated and observed flow match is that if we add a net circulation to the flow. This circulation effectively moves the point where the streamlines connect with the solid airfoil. The behavior with and without circulation is shown in Figure 12.7 for a flat plate airfoil.

This seems arbitrary? Why would there be net circulation? The source of the circulation is subtle. Imagine we start an airfoil from rest. We draw loop around the airfoil and since everything is at rest, the circulation is zero. If we impulsively start an airfoil, we would see that a vortex gets shed from the airfoil at the instant that it starts. This is just like moving the spoon in the water. However, the circulation theorem would tell us that the circulation around the loop is still zero. Therefore, this starting vortex would need to be compensated by some other circulation of opposite sign. This is the circulation around the airfoil. When we impulsively start the airfoil, viscosity acts to move the point where the streamline separates from the airfoil. In doing so,



a) b)

Figure 12.7 A flat plate airfoil in potential flow. In (a) there is no circulation and streamlines wrap around the rear of the airfoil. In (b) there is circulation and the rear streamline leaves the airfoil at the trailing edge.

vorticity is created and the counteracting circulation around the airfoil must keep the net circulation at zero.

While the starting vortex shows that circulation must exist around the airfoil, the idea that the circulation must exactly the amount that moves the streamline to the trailing edge is not something that we can prove. It is just a useful rule of thumb that turns out to work as long as the airfoil is not at too steep of an angle of attack. This rule of thumb is known as the Kutta condition. Once you know the flow field from the solution of Euler's equation, you can find the pressure from the fact that in the absence of gravity effects Bernoulli holds everywhere, $P+\frac{1}{2}\rho(\nabla\phi)^2 = \frac{1}{2}\rho U_0^2$. Once you know the pressure, you you can find the lift on the airfoil by integrating the pressure around the surface of the airfoil and it turns out to actually agree pretty well with experiments.

For the same dimensional analysis arguments as with drag, when we study lift we define the coefficient of lift in a similar way,

$$C_L = \frac{\text{Force/width}}{\frac{1}{2}\rho U^2 \ell},$$

where ℓ is the length of the airfoil (which must be precisely defined for any shape that is not a flat plate). If we dig into the mathematical theory of potential (or irrotational) flow, it turns out we don't need a computer to solve for the flow field around a flat plate inclined to the flow. The analysis (which you can find in a number of books if you are

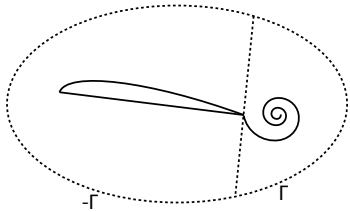


Figure 12.8 Schematic of the starting vortex for an airfoil. The circulation calculated around the outer oval must be zero, so the circulation around the two inner loops must cancel each other out. The airfoil must have circulation of the opposite sign as the starting vortex. If the airfoil is suddenly stopped a vortex of the opposite sign as the starting vortex is shed in order to maintain no net circulation.

interested) yields a surprisingly simple formula,

$$C_L = 2\pi \sin \alpha,$$

where α is the angle the airfoil makes with the flow field, called the angle of attack. This simple theoretical results compared to experimental data is shown in Figure 12.9a. In the figure we see good agreement. At higher angles we observe flow separation and a complex wake behind the airfoil - the simple inviscid theory fails. At a high angle of attack, flow separation can be so severe as to lead to stall.

You may have noticed that most planes you fly don't have flat plates for wings. In 1933 NACA (a predecessor of NASA) concluded a large study of measuring lift and drag coefficients for numerous airfoil shapes. A sample of that orignal data for a single airfoil is shown in Figure 12.9b. The figure shows the coefficient of lift, coefficient of drag, the ratio of lift to drag (L/D), and the center of pressure (c.p.) as a function of angle of attack. It might take a little study to read the plot. However, if you look carefully you can see that the lift coefficient is well described at modest angles of attack by the formula $C_L = 2\pi \sin(\alpha)$. Note that at low angles of attack the drag coefficient is very small. The measurement behavior is pretty close to what Euler equations predict; i.e. no drag.

It turns out that there is a mathematical tricks that allows us to compute the full velocity field for different airfoil shapes in potential flow. The trick allows us to transform the solution for flow around a cylinder to flow around different airfoil shapes. Called conformal map-

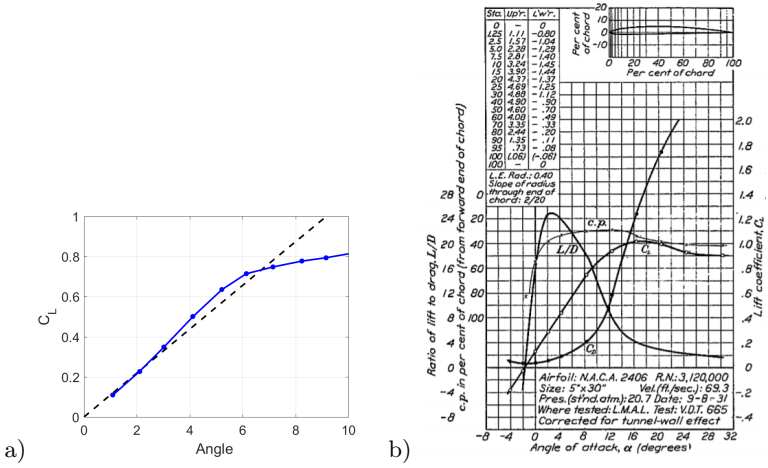


Figure 12.9 a) Coefficient of lift from experiments for an inclined flat plate compared to the single equation from irrotational flow theory. Data from "Low speed airfoil data" by Williamson et al. b) Lift and drag data from the original 1933 NACA study for a single airfoil.

ping, if we have a mathematical expression to squish a cylinder to a new shape, we can use the same expression to squish the solution too (I am oversimplifying things of course). While conformal mapping is well beyond what we would cover in a first course, it is interesting to note that airfoil theory is one area where our potential flow solutions work quite well and a fully analytical approach provides useful predictions.

13
Boundary layers

The analysis of fluid flow via Euler's equations seems relatively nice. While we presented only a few highlights in this book you should have gotten the impression that a number of problems could be readily solved mathematically or computationally using Euler's equations for incompressible flow. So while we know viscosity is always with us, it is a small force compared to the others in the problem in a number of applications. At the scale of humans, everything is high Reynolds number. However, it has been observed for a long time that Euler's equations do an unusually bad job at making useful predictions. Euler's equation predicts no drag force, for example. It was not until the early 1900s when Prandtl introduced the idea of the boundary layer, that people began to appreciate why Euler's equations were so poor and how to rectify our understanding (Anderson (2005)).

If we have a high Reynolds number flow around a streamlined object such as an airfoil, what is observed is that the fluid velocity just away from the surface of the airfoil very closely matches the solution to Euler's equations. This part of the flow is not influenced by viscosity. As we approach the surface, the flow velocity goes to zero because the no-slip condition is obeyed. However, the size of this *boundary layer* is extraordinarily thin. Flow over an airplane wing could have boundary layers measured in the units of millimeters. Why should this thin region play such a major role in determining flow? The answer is the topic of this chapter.

We need to keep in mind all the forces at play in our problem. When we looked at solutions to the Navier-Stokes equations we found we could readily solve problems where there was no fluid acceleration - the

balance was between viscosity and some driving force such as applied pressure. When we discussed Bernoulli's and Euler's equation, we consider the balance between pressure and acceleration. In this chapter we will find that in many cases there is a complex interplay of fluid acceleration, viscosity and pressure. While our analytical tools get more complex as the flows do, all is not hopeless and there is a lot we can do with simple problems.

13.1 Impulsively started plate

Let's start with an idealized problem that has an analytical answer to begin to explain this boundary layer. Imagine a flat plate that is infinite in extent in the x direction. The plate and the fluid above it are initially at rest at $t = 0$. The fluid above the plate extends to infinity. At $t = 0$, the plate is impulsively started to move with a constant velocity U. You should turn back to the chapter on Navier-Stokes solutions and look at the problem of transient Couette flow. The stated problem here is the same, only in the Couette flow the domain was bounded in the y direction whereas here the domain extends infinitely in y. Since the two problems are so similar, the development of the reduced form of the Navier-Stokes is identical as transient Couette flow.

The reduced form of the Navier-Stokes in terms of the kinematic viscosity, ν, is the 1D heat equation,

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2}.$$

Recall that the kinematic viscosity plays the same role in this equation as thermal diffusivity in the heat equation. We expect that the behavior would be the same as applying an instantaneous change in temperature to a semi-infinite solid.

We now want to make this equation dimensionless as we did with Couette flow. We can set the velocity scale to $\tilde{u} = u/U$. However, what should the time scale or spatial scale be? There is no natural geometric length to scale length by. Our only choice is δ, the thickness of the fluid region which is in motion. However this is an unknown scale. Using an

arbitrary length $\tilde{y} = y/\delta$ and $\tilde{t} = t/t_0$ would yield,

$$\frac{\partial \tilde{u}}{\partial \tilde{t}} = \frac{\nu t_0}{\delta^2} \frac{\partial^2 \tilde{u}}{\partial \tilde{y}^2}.$$

This scaling indicates that the thickness of the region where the flow transitions from the plate velocity, to being at rest grows with the square root of time and depends on the kinematic viscosity of the fluid,

$$\delta = \sqrt{\nu t}.$$

This scaling of the growth of the viscous layer what we expect for all diffusion problems - in the boundary layer it is diffusion of momentum.

The scaling argument is confirmed by the analytical solution to this problem, which we will not derive but has the form,

$$\frac{u(y,t)}{U} = \left(1 - \text{erf}\left(\frac{y}{2\sqrt{\nu t}}\right)\right).$$

where erf is known as the error function. If you are unfamiliar with the error function, you can look up it's definition and you will find that most mathematical software will understand the definition and generate a plot of the function for you. The dimensional solution is shown in Figure 13.1. Since $\nu = 10^{-6}$ m^2/s in water, you can see that the length scale of the boundary layer is pretty small. If we pulled the plate for an hour in water, the thickness of the boundary layer would be 6 cm.

What is interesting about the analytical solution is that the solution is a single function of a variable $y/(2\sqrt{\nu t})$ which mixes the spatial and time dependence of the problem into a single variable. This feature is called a similarity solution. Similarity solutions are common in diffusion problems where there is no natural length scale. In this problem the length scale for the problem is the boundary layer thickness, $\sqrt{\nu t}$, which depends on time. What is interesting about the solution is that the shape of the boundary layer remains the same, it just gets thicker and thicker with time.

A somewhat obvious but crucial point here is that the boundary layer will exist regardless of the smallness of viscosity, as long as it is not zero. Unless you are working with liquid helium near zero kelvin and approach an exotic state known as the superfluidity, all fluids have some viscosity. So even if viscosity is small, the boundary layer still exists. If I am close to the surface I cannot ignore viscosity.

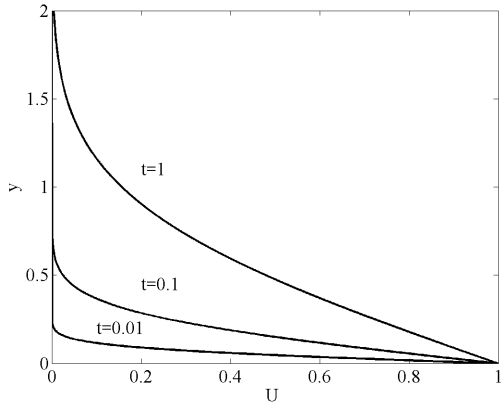


Figure 13.1 Solution to the problem of an impulsively started plate at different instances in time. Here we set $\nu = 1$. If the vertical axis is plotted as $y/(2\sqrt{\nu t})$, then there is a single curve.

13.2 Boundary layer equations and laminar solution

Imagine flow over a stationary flat plate, a problem similar but different from the previous section. The flow approaches the plate with uniform flow velocity, U_∞. The plate is flat and thus far from the plate we expect the flow not to be disturbed. At the surface of the plate, the velocity must go to zero due to the no-slip condition. What is observed is that a boundary layer region grows from the leading edge of the plate. The boundary layer is simply the transition region where the velocity goes from zero at the plate surface to the free stream value. The schematic of the observed boundary layer is shown in Figure 13.2.

If we go to dimensional analysis and the Pi Theorem to ask what is the boundary layer thickness, δ, at the end of a plate of length L. The boundary layer thickness δ could then depend upon variables L, U_∞, μ and ρ. The Pi Theorem would tell us that there are 5 variables expressed in 3 dimensions so there should be only two dimensionless parameters. The dimensional analysis would yield,

$$\frac{\delta}{L} = f\left(Re\right)$$

where the Reynolds number based on the plate length is $Re_L = \rho U_\infty L/\mu$.

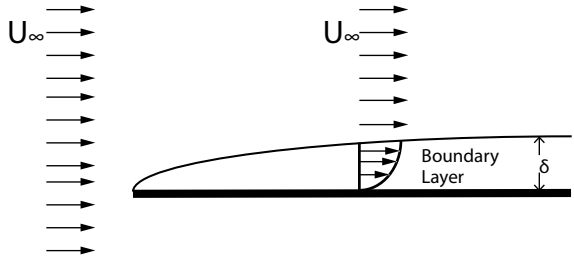


Figure 13.2 Schematic of the laminar boundary layer which develops as a uniform flow approaches a flat plate.

The boundary layer thickness can only depend upon the plate Reynolds number.

We could estimate the dependence by a simple argument using the solution to the impulsively started plate, $\delta \sim \sqrt{\nu t}$. In the boundary layer problem, there is no time. At a given downstream location, the "time" is the time it takes fluid to reach that point, i.e. $t \sim L/U_\infty$.

$$\delta \sim \sqrt{\nu t} \sim \sqrt{\frac{\nu L}{U_\infty}}.$$

In terms of the Reynolds number this equation can be rewritten as

$$\frac{\delta}{L} \sim \sqrt{\frac{\nu}{U_\infty L}} \sim \sqrt{\frac{1}{Re}}.$$

Since the flow progresses downstream, we the argument above works for any x location. Thus we might expect,

$$\frac{\delta}{x} \sim \sqrt{\frac{\nu}{U_\infty x}} \sim \sqrt{\frac{1}{Re_x}},$$

where we use the Reynolds number based on the x location. This scaling suggests that the boundary layer thickness grows as the square root of the distance down the plate. To get a sense of scale, lets take air $\nu = 10^{-6}$ flowing at $U_\infty = 1$ m/s over a 1 m length plate. The boundary layer would be 1 mm. If the velocity were increased to 100 m/s (an airplane taking off), the boundary layer thickness would be 100 microns.

The shear stress on the plate due to viscosity can also be estimated. For a Newtonian fluid, you can look back at the previous chapter and

see that the shear stress (tangential to the surface) is $\tau = \mu \partial u/\partial y$. The stress is given by the velocity gradient at the surface. The shear stress would be approximately,

$$\tau(x) = \mu \frac{\partial u}{\partial y} \sim \mu \frac{U_\infty}{\delta} \sim \mu \frac{U_\infty}{x\delta/x} \sim \frac{\mu U_\infty}{x} \sqrt{\mathrm{Re}_x}$$

Making an analogy with drag, we would follow dimensional analysis to define the coefficient of friction as,

$$C_f = \frac{\tau(x)}{\rho U_\infty^2/2}.$$

Using the estimate we have already obtain for τ we have an estimate for the scaling of the coefficient of friction,

$$C_f \sim \frac{\mu U_\infty}{x\rho U_\infty^2/2} \sqrt{\mathrm{Re}_x} \sim \frac{2\mu}{x\rho U_\infty} \sqrt{\mathrm{Re}_x} \sim \sqrt{\frac{1}{Re_x}}.$$

This simple estimate predicts that the shear stress should be highest near the leading edge of the plate and continually decrease with downstream distance. The total drag force, F, for a plate of length L, is then calculated from $F = \int \tau(x) dx$. The total drag coefficient is thus given as,

$$C_D = \frac{F}{L\rho U_\infty^2/2} \sim \frac{1}{\sqrt{\mathrm{Re}_L}}.$$

From such simple arguments we have no way to compute what the pre-factors in these estimates might be. However, we find that we can get the basic scaling and qualitative behavior for the laminar boundary layer.

13.2.1 Full solution to the boundary layer

The full boundary layer problem over a flat plate has a mathematical solution. This solution makes use of the fact that δ/x is usually a pretty small number. The basic idea is that when there is a vast difference in scales, one can make the equations dimensionless using the different scales. So we will scale the x coordinate by the length of the plate, L, and the y direction is scaled by $\delta = \sqrt{\frac{\nu L}{U_\infty}}$. We will also use different scales for the u and v velocities; $\tilde{u} = u/U_\infty$ and $\tilde{v} = v/V_0$.

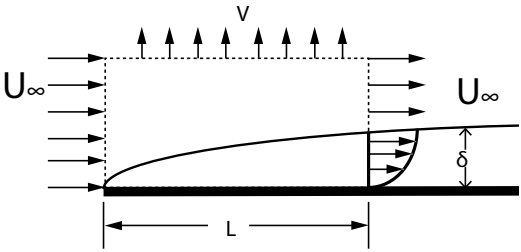


Figure 13.3 Schematic of control volume for the boundary layer problem. The flow into the left side of the control exits from the right side with a deficit on the order of $U_\infty \delta$. The extra flow that came through the left side must exit the upper surface of the control volume. The magnitude of this flow is on the order of $V_0 L$.

Conservation of mass gives.

$$\frac{U_\infty}{L}\frac{\partial \tilde{u}}{\partial \tilde{x}} + \frac{V_0}{\delta}\frac{\partial \tilde{v}}{\partial \tilde{y}} = 0$$

This equation implies that $V_0 = U_\infty \delta / L$ would be a good choice. This scaling implies that the vertical velocities are less than the horizontal velocities by a factor of δ/L. If you think about this comment for a minute, it should make some physical sense to you. The schematic in Figure 13.3 shows a control volume picture of the scaling.

Applying this scaling for V_0 and making the equation dimensionless would yield,

$$\left(\tilde{u}\frac{\partial \tilde{u}}{\partial \tilde{x}} + \tilde{v}\frac{\partial \tilde{u}}{\partial \tilde{y}}\right) = -\frac{\partial \tilde{P}}{\partial \tilde{x}} + \frac{\mu}{\rho U_\infty L}\left(\frac{\delta^2}{L^2}\frac{\partial^2 \tilde{u}}{\partial \tilde{x}^2} + \frac{\partial^2 \tilde{u}}{\partial \tilde{y}^2}\right)$$

and

$$\frac{\delta^2}{L^2}\left(\tilde{u}\frac{\partial \tilde{v}}{\partial \tilde{x}} + \tilde{v}\frac{\partial \tilde{v}}{\partial \tilde{y}}\right) = -\frac{\partial \tilde{P}}{\partial \tilde{y}} + \frac{\delta^2}{L^2}\frac{\mu}{\rho U_\infty L}\left(\frac{\delta^2}{L^2}\frac{\partial^2 \tilde{v}}{\partial \tilde{x}^2} + \frac{\partial^2 \tilde{v}}{\partial \tilde{y}^2}\right).$$

Since δ/L is a small number and we have scaled all the partial derivative terms in our equation to be on the order of magnitude of 1, we can take the perhaps somewhat uncomfortable step of removing all terms with small numbers out in front. This step seems odd the first time you see it. However, it is pretty standard practice. It is a logical thing to do.

We have supposedly scaled all the partial derivatives in the equation to be on the order of 1. Thus the terms with $(\delta/L)^2$ simply go away because they are small. Taking this leap of faith provides the boundary layer equations,

$$\tilde{u}\frac{\partial \tilde{u}}{\partial \tilde{x}} + \tilde{v}\frac{\partial \tilde{u}}{\partial \tilde{y}} = -\frac{\partial \tilde{P}}{\partial \tilde{x}} + \frac{1}{Re}\frac{\partial^2 \tilde{u}}{\partial \tilde{y}^2} \tag{13.1}$$

and

$$0 = \frac{\partial \tilde{P}}{\partial \tilde{y}}. \tag{13.2}$$

The second equation simply states that the pressure across the boundary layer is a constant. The fact that the pressure across the boundary is constant is really useful. What you can do, in practice, is solve Euler's equations for flow around your object. This solution gives you the pressure at the surface of the object. You can assume that since the boundary layer is so thin that the pressure at the surface of the object and the pressure at the edge of the boundary layer would be the same. Thus, you get the pressure and $\frac{\partial \tilde{P}}{\partial \tilde{x}}$ from the solution to Euler's equation, and then use the now known pressure gradient to solve the boundary layer equations, Equation 13.1.

In the flat plate problem, the pressure "far away" from the plate (in units of δ) is just a constant and there is no pressure gradient. Thus, our final boundary layer equations for the two components of velocity are,

$$\tilde{u}\frac{\partial \tilde{u}}{\partial \tilde{x}} + \tilde{v}\frac{\partial \tilde{u}}{\partial \tilde{y}} = \frac{1}{Re}\frac{\partial^2 \tilde{u}}{\partial \tilde{y}^2}$$

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

The boundary layer equations describe a balance between viscosity and fluid acceleration.

There is a mathematical solution to this flat-plate problem, and we will skip the details and only show the result. The analytical solution is not really a closed form answer. The mathematical solution simply transforms this partial differential equation into an equivalent ordinary differential equation. This ODE still needs to be solved numerically. The solution to the velocity profile is shown in Figure 13.4.

Once the boundary layer equations are solved numerically, you can

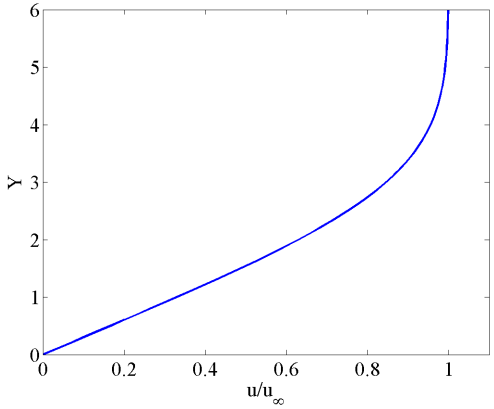


Figure 13.4 Solution to the laminar boundary layer problem over a flat plate. The figure shows the universal velocity profile near the plate. The y axis variable is $y\sqrt{\frac{U_\infty}{x\nu}}$.

calculate everything you want to know about the boundary layer. The thickness, defined as the distance when the velocity field is 99 % of the free stream is,

$$\frac{\delta}{x} = \frac{5}{\sqrt{\mathrm{Re}_x}}.$$

The factor of 5 comes from the solution, but the form was acquired through dimensional arguments. The shear stress vector at a point in a 2D flow is given by $\mathbf{n} \cdot \mathbf{T}$. We can evaluate the stress at the point to be in dimensionless terms,

$$C_f = \frac{\tau(x)}{\rho U_\infty^2/2} = \frac{0.664}{\sqrt{\mathrm{Re}_x}},$$

where C_f is known as the coefficient of friction. The total drag force (per unit width) is thus given as,

$$C_D = \frac{F}{L\rho U_\infty^2/2} = \frac{1.328}{\sqrt{\mathrm{Re}_L}}.$$

The numerical values come from the calculated numerical solution. The really important point is that the basic scaling of the answer comes from consideration of dimensional analysis and some physical arguments.

13.3 Turbulent boundary layers

Any flow at high Reynolds number is subject to becoming unstable and turbulent. If the boundary layer has become turbulent, there is no mathematical solution to the problem and the laminar solution is invalid. There is an extraordinary amount of theoretical, experimental, and numerical work on the problem of the turbulent boundary layer. The reason for all the interest is that the problem is important. Turbulent boundary layers are seen in many applications and they play a strong role in determining drag on boats, planes, and cars and thus determines the energy use and efficiency of transportation. Boundary layers also occur in the atmosphere and play a strong role in determining the weather and climate. It is difficult in a short introduction like this to convey the importance of the turbulent boundary layer on many problems of practical importance. There is no way I can adequately convey both the amount that is known and unknown about turbulent boundary layers.

The rule of thumb is that for a flat plate, the local Reynolds number, $Re_x = \rho U x/\mu$ should be

$$Re_x < 10^5 \quad \text{for laminar boundary layers.}$$

In this definition, the local Reynolds number increases with distance down the plate. Thus all boundary layers start laminar. For a sense of scale, a 1 m/s flow in water would go turbulent around 10 cm from the leading edge. The criterion for laminar flow is not an exact number. It is a rule of thumb. The exact location of turbulence can be very difficult to predict and is very sporadic, just like we have discussed in other applications such as pipe flow. The exact location depends upon whether an experiment is carefully done to remove all perturbations or whether we are in a "real world" application. For a carefully conducted experiment where the upstream flow is very stable and you are careful to make a perfectly smooth plate, the transition to turbulence occurs at a higher Reynolds number.

Once the boundary layer is turbulent, we make use of the fact that dimensional analysis tells us that

$$\frac{\delta}{x} = f\left(Re_x\right).$$

Thus a small number of experiments can provide an empirical solu-

tion to the boundary layer thickness. Empirical fits to the data given boundary layer thickness as,

$$\frac{\delta}{x} = \frac{0.37}{\mathrm{Re}_x^{1/5}},$$

and the coefficient of friction as

$$C_f = \frac{0.0592}{\mathrm{Re}_x^{1/5}}.$$

These are not fundamental laws and there is no mathematical derivation, the expressions just fit the data well. Notice that the growth of the layer with distance down the plate is

$$\delta \sim x^{4/5}$$

which is much more rapidly than with the laminar boundary layer.

The turbulent velocity profile is thicker, but much blunter than the laminar one. The random turbulent eddies pull high momentum fluid from the free stream and bring it close to the solid surface. The viscous drag is much higher in turbulent flow due to the steep velocity gradients at the surface. The high momentum fluid sits closer to the plate as shown in Figure 13.5a. So while the turbulent layer is thicker than the laminar one, the velocity gradient at he surface is much steeper. The boundary layer thickness as a function of distance down the plate are shown in Figure 13.5b. There is a jump at $Re = 10^5$ in the two solutions. In reality, there is not a jump but a transition regime from one solution to the other. The thickness of the boundary layer rapidly expands when the boundary layer becomes turbulent. Because the velocity gradient on the blunt turbulent velocity profile is greater than the laminar one, the shear stress at the solid surface is greater for the turbulent boundary layer.

13.4 Boundary layer separation

Now consider 2D flow over a cylinder. Euler's equations would predict streamlines that approach and leave the centerline. The flow would have fore-aft symmetry as shown in Figure 13.6a. This is not what is observed. What is observed is an increasingly complicated asymmetry. The asymmetry starts with the flow behind the cylinder to have

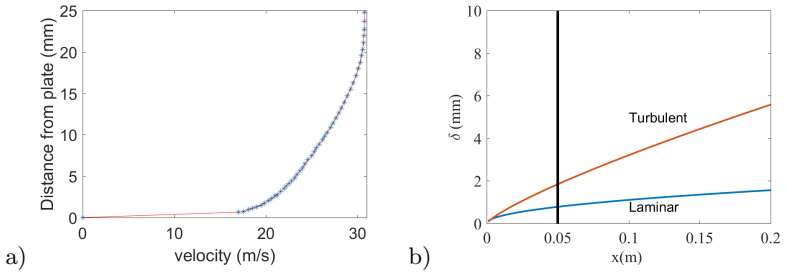
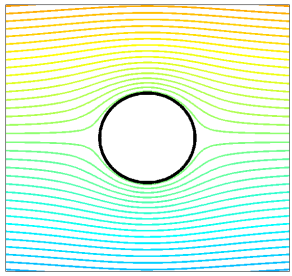


Figure 13.5 a) Experimental data for a turbulent boundary layer in air. The data is openly provided by Princeton University Gas Dynamics lab. Note the very steep velocity gradient at the plate surface b) Boundary layer thickness as a function of distance down the plate from the empirical correlations. The jump from the lower laminar solution to the upper turbulent one would occur around the location of the vertical line. For $x < 0.05$ we observe the laminar boundary layer solution, for $x > 0.05$ m we observe the turbulent one. In reality the transition happens over some distance and is not a vertical jump.

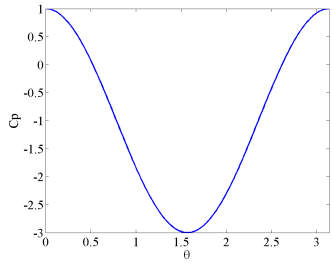
some small vortices or recirculation. As the flow speed is increased, an increasingly complicated structure of the wake emerges. What we are seeing in the complex wake is a phenomena called boundary layer separation.

The boundary layer equations derived in the previous section provide a useful and practical way to calculate the flow using Euler's equations and the boundary layer equations. The basic idea is that you solve Euler's equation for an arbitrary shaped object. This solution provides the pressure everywhere along the surface. Since the pressure is constant *across* the boundary layer, Euler's equation provides $\partial P/\partial x$ in the boundary layer equation.

Let's apply this idea to flow over a cylinder. If we think about the pressure, Bernoulli's (Euler) equation would tell us that the pressure at the stagnation point at the front side of the cylinder would be greater than the ambient pressure by an amount $\frac{1}{2}\rho U^2$. Since the flow is symmetric by Euler's equation, the pressure at the rear stagnation point is also higher than the ambient. At the top and bottom of our cylinder, the pressure is low. The velocity will speed up to scoot around the cylinder. Thus the pressure at the top and bottom are low. The pressure along the surface of the cylinder as predicted by Euler's equations



a)



b)

Figure 13.6 a) Streamlines for irrotational, inviscid flow over a cylinder. b) The pressure along the surface of the cylinder as a function of angle. The data are plotted as the coefficient of pressure which is defined as $Cp = \frac{P-P_\infty}{\frac{1}{2}\rho U_\infty^2}$. The pressure at the forward and rear stagnation points is higher than the ambient pressure while the pressure at the top of the sphere is the lowest. The forward and rear stagnation points are at 0 and π.

are shown in Figure 13.6. The pressure distribution as predicted by the equations follows our simple qualitative picture.

As flow reaches the front of the cylinder and moves toward the top, the pressure gradient in the boundary layer, dP/dx, is such that the flow is going from a region of high to low pressure. We saw the pressure gradient is favorable in that it is pushing with the direction of flow. As the flow proceeds from the top of the cylinder toward the rear stagnation point, the pressure is *increasing* with the flow direction. We saw the pressure gradient is adverse as it pushes against the flow. When solving Euler's equation this adverse pressure gradient is perfectly balanced by the momentum of the fluid; the pressure gradient slows the fluid down. However, the fluid near the surface of the cylinder, inside the boundary layer, is deprived of its momentum due to the viscous boundary layer. The adverse pressure gradient pushes back on the fluid in the boundary layer and the fluid in this region has insufficient momentum to overcome the adverse pressure gradient. This adverse pressure gradient can cause reverse flow in the boundary layer on the back side of the cylinder. This reverse flow is unstable and creates what is called boundary layer separation. This separation can be seen in the experimental image in Figure 13.7.

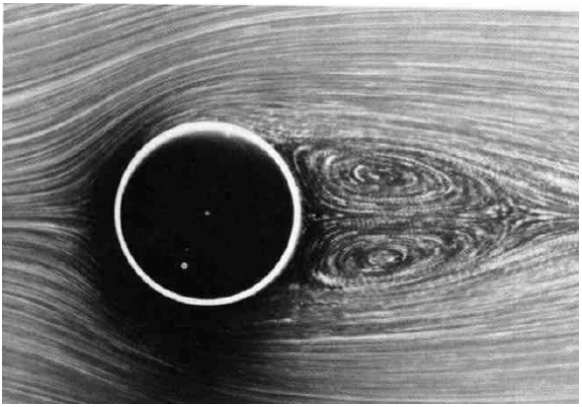


Figure 13.7 Boundary layer separation for flow over a cylinder (From Album of Fluid Motion). The Reynolds number is 26. For Reynolds numbers less than 1, the flow shows fore-aft symmetry.

13.5 Observations of drag on a sphere

Lets reconsider the problem of drag on a sphere moving at constant speed. In theory we can solve the Navier-Stokes equations to obtain the velocity field and pressure at every point. In practice we can create the solution numerically using CFD software. Once the velocity and pressure is known, we can compute the net force on the sphere by integrating the stress vector around the surface of the sphere. The stress vector is the dot product of the normal vector and the stress tensor, $\mathbf{n} \cdot \mathbf{T}$. The force is found by the the surface integral of the stress,

$$\mathbf{F} = \int \mathbf{n} \cdot (P + \mu(\nabla \mathbf{v} + (\nabla \mathbf{v})^T))dS$$

Note that the total force has a contribution from pressure and viscous stresses - distinguishing between these two forces will be important to our understanding soon. From dimensional analysis, we know that the drag force is expressed as the drag coefficient which is only a function of the Reynolds number

$$\frac{F}{\frac{1}{2}\rho U_0^2 \pi R^2} = C_d(\mathrm{Re}).$$

Knowing that there is some functional relationship for the drag coef-

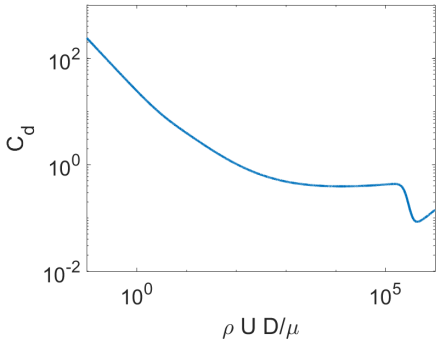


Figure 13.8 Drag coefficient as a function of Reynolds number for flow around a sphere.

ficient as a function of Reynolds number allows us to collapse experimental results for all sphere sizes at all velocities and in all fluids onto a single master curve, Figure 13.8.

Now that we know a little bit about boundary layers, let's look more closely at the data in Figure 13.8. At low Reynolds number there is an analytical solution which provides $C_D = 24/\mathrm{Re}$ - a solution supported by the experiments. As the Reynolds number is increased, the drag coefficient limits to a constant of approximately 0.4. The low Reynolds number solution is lost once boundary layer separation begins to set in. These two behaviors were previously explained as the transition from viscosity dominated flow to inertia (or density) dominated flow. As the Reynolds number is further increased about 10^5, the drag coefficient goes through a sudden decrease. This dip is very unexpected. We might think that as the flow increases, the boundary layer would eventually become turbulent and the drag should go up. Why did it instead decrease?

The reason is that the total drag on the sphere is given by pressure drag and friction (or viscous drag) along the surface. You already know about different types of drag, even if you don't know that you do. Next time you are in the car, put your hand out the window. Turn it horizontal with the ground and then turn it so it is vertical to the ground. Feel the difference? The difference you feel is pressure and friction. On a flat plate (your horizontal hand), there is only friction at

the surface due to the velocity gradient. The shear stress at the surface depends upon the viscosity of the fluid. On a sphere, the stress vector at a point is given by the velocity gradient at the surface and the local pressure. In Euler's equation the pressure on the surface of the sphere is symmetric, thus the net force due to pressure is zero. If we watch the experimental streamlines at high Reynolds number where the boundary layer around the sphere is still laminar but strong separation sets in, we would see the streamlines move around the sphere, but do not recover on the other side. If we measured the pressure on the surface we find strong asymmetry when there is flow separation. Thus pressure at the forward stagnation point is greater than the ambient by $\rho U^2/2$. On the back side of the sphere, the wake is relatively quiet. If we look at the streamlines at the top or bottom of the sphere, they would be straight and thus the pressure would be the ambient. Thus the pressure in the wake should be something close to the ambient pressure. Therefore, very crudely, the force acting on the front half of the sphere due to pressure is $\frac{1}{2}\rho U^2 \pi r^2$. The force in the back half is zero. Thus the net force due to *pressure drag* is on the order of $\frac{1}{2}\rho U^2 \pi r^2$. Thus the drag coefficient is about 1. Pressure drag dominates over friction at the surface in this regime.

However, we said that separation occurs because momentum is deprived from the fluid in the boundary layer. When the boundary layer becomes turbulent, the velocity profile gets blunter. The transition from zero velocity at the wall to the free stream value occurs over an even smaller distance than the laminar case. Thus, high momentum fluid is even closer to the wall. The high momentum fluid near the wall allows the turbulent boundary layer to push into the adverse pressure gradient further than the laminar counterpart. Separation is delayed and occurs further down the back of the sphere. This effect can be seen in the experimental images in Figure 13.9. The low pressure wake acts over a smaller area when the boundary layer is turbulent rather than laminar. Pressure drag is decreased when the boundary layer is turbulent. So even though the steeper velocity gradients in a turbulent flow increase the friction at the surface, the net drag is dominated by pressure drag and thus the overall drag force (coefficient) decreased when the boundary layer becomes turbulent. The pressure drag is reduced due to the delayed separation. Since the pressure drag dominates over the

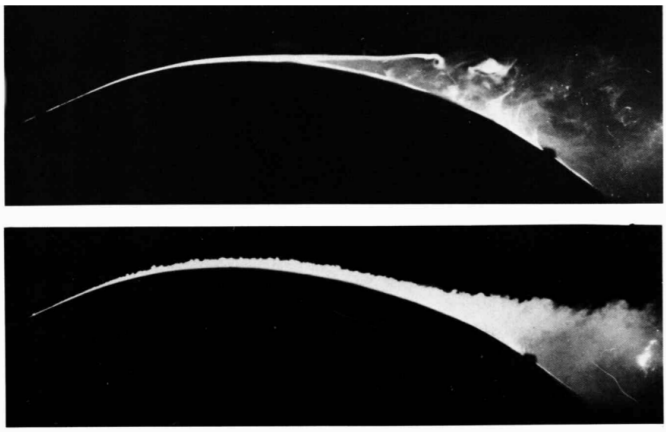


Figure 13.9 Boundary layer separation for a laminar (upper picture) and turbulent (lower picture) boundary layer. The delay of separation is clearly seen in the case of the turbulent boundary layer.

viscous drag, the net effect is that the turbulent boundary layer has lower overall drag.

The role of boundary turbulence in reducing drag is the reason for dimples on golf balls. The irregular surface induces turbulence to occur at lower Reynolds number than a smooth sphere. The turbulent boundary layer delays separation and the golf ball goes further. Imperfections on the surface of a sphere and its effect on tripping the boundary layer into a turbulent state can help explain why curveballs curve in baseball.

14
Turbulence

As we have repeatedly stated, the equations of fluid dynamics are nonlinear and challenging to solve. One of the key challenges is that many of the engineering flows we are interested in are high Reynolds number, and thus turbulent. Over the past century there have been countless researchers who have devoted their careers to trying to "solve" the turbulence problem. The first studies of turbulence are often credited to Leonardo Da Vinci, who has a number of sketches of turbulent flows in his work. Nobel physicist Richard Feynman called turbulence one of the most important unsolved problems of classical physics. Even with all this effort there are perhaps even more questions than answers. I do not mean to say that because the problem is not "solved" everything is hopeless and unknown. There have been many modern experimental and theoretical advances in physics, mathematics, and engineering that have come as the result of all this effort. I simply want to stress that this is a very complicated topic and my treatment in a short chapter is going to be embarrassingly incomplete. There are a few concepts you should know, that is the point of this chapter.

14.1 Can we simulate turbulence?

There is nothing about turbulence that is not embodied in the Navier-Stokes equations. In theory we could just get a computer to solve the Navier Stokes equations numerically and solve for all the details. This approach is called Direct Numerical Simulation (DNS) and it is feasible if the Reynolds number is not too large. However, turbulence can chal-

lenge the fastest supercomputers and we can quickly exceed the limits of the worlds fastest computers. For many problems, the solutions span too many orders of magnitude in time and space to be resolved. Further, the outlook is such that for many important problems waiting for computers to get faster doesn't seem like a good prospect for the near future, even if Moore's law continues to hold.

The problem is that scaling arguments show that the total amount of computational effort to solve a 3D turbulent flow scales very poorly with Reynolds number. The total amount of computational effort scales as

$$\text{total computing power} \sim \mathrm{Re}^3.$$

To get one order of magnitude increase in Reynolds number we need 1000 times the computing power. If you look at curves of supercomputer performance over the years, we historically get about a factor of 1000 improvement over 10 years. A similar though perhaps a bit slower time scale holds for desktop computers. This fact is actually quite remarkable, making things that were cutting edge when I was a graduate student trivial for today's computers. However, for something like atmospheric flows we are perhaps 10 orders of magnitude in Reynolds number or more off in what we can calculate and what we would like. In Figure 14.1 is an image of a water jet at relatively low Reynolds number, yet the complexity and diverse length scales are readily apparent. Even by the most optimistic estimates it will be at least several decades before a full DNS of the atmosphere scale is feasible.

You might be tempted to simply truncate the size a DNS and solve for only the large features. If you are interested in the dominant features of the climate do you really need to resolve everything? If we are trying to model the weather, can't we make the grid scale on the order of several miles such that we don't capture every detail but the dominant motions? The reason you cannot simply truncate the simulation is due to the energy cascade, which we now explain.

14.2 Kolmogorov theory and the energy cascade

If we consider homogeneous turbulence, turbulence that is the same in all directions like a big box of shaken up fluid, we can make some

Figure 14.1 Experimental image from a water jet at Reynolds number ~ 2000. Even at this Reynolds number, the complexity in the number of length scales seen in the flow is striking. From Album of Fluid Motion.

progress by considering only dimensional analysis and a simple model. This analysis put forth by Kolmogorov in 1941 remains a centerpiece of turbulence theory.

The classic picture of turbulence is that of the energy cascade. The flow is considered to consist of a number of turbulent eddies of different sizes. The model is that the largest eddies split into smaller eddies, which split into smaller eddies, which split into even smaller eddies. Every time the eddies split the kinetic energy is equally divided into new, smaller sized eddies. There is no loss in this process. The cascade continues to smaller and smaller scales until finally the local Reynolds number is sufficiently small that viscosity can take the energy away. The energy at the smallest scales are turned into heat and the kinetic energy vanishes. This cascade of energy can occur over several orders of magnitude in length.

From Kolmogorov's scaling arguments it turns out that we could estimate the length scale where viscosity removes energy from the system, η, as

$$\frac{L}{\eta} \sim \mathrm{Re}^{3/4}$$

where L is the length scale of the largest eddies and the start of the cascade. For an atmospheric flow we can easily reach Reynolds numbers of 10^{12} or higher; there are indeed many orders of magnitude of length over which the turbulence cascade exists.

We can describe the flow via a spatial Fourier series $u(\mathbf{x}) = \sum A_k e^{i\mathbf{kx}}$, where the wavenumber $\mathbf{k}$ is like a spatial frequency. Kolmogorov argued from physical arguments and dimensional analysis that the energy per wavenumber follows a scaling,

$$E(k) \propto k^{\frac{-5}{3}}$$

where

$$\text{Total kinetic energy} = \int E(k)dk.$$

This predicted energy spectrum comes from simple arguments of units and a little physical interpretation. Note that as k increases the length scale decreases. The picture is there is an inertial range where viscosity is not important and the energy in the fluid flow follows the Kolmogorov scaling. Energy progresses down the $k^{\frac{-5}{3}}$ cascade until such large wavenumbers (small length scale) that viscosity becomes important and the scaling no longer holds. This energy spectrum has been measured and generally experiments agree quite well with this simple scaling law, as seen in Figure 14.2.

Imagine a bucket of water that you are stirring constantly with a big spoon. Energy is constantly being put into the system with a length scale on the size of the stir (let's say that is the size of the bucket). The big swirls you create break apart and that energy goes into smaller and smaller swirls. This cascade of energy from large scales to small scales follows the Kolmogorov scaling. Once the eddy size gets small enough that viscosity becomes important the eddy motion is halted and the energy goes into heating the fluid. You reach a steady state where the energy you constantly put into the system at the large scale is turned into thermal energy at the small scale. Just to give some numbers to this problem, let's say our bucket is a foot in diameter (≈ 30 cm) and we are stirring at 1 diameter per second (30 cm/s). This yields a Reynolds number of Re = 90,000 and a dissipation length scale is about $\eta = 50$ microns. Thus our inertial cascade spans nearly 4 orders of magnitude.

The picture of turbulence presented here, hardly captures all the

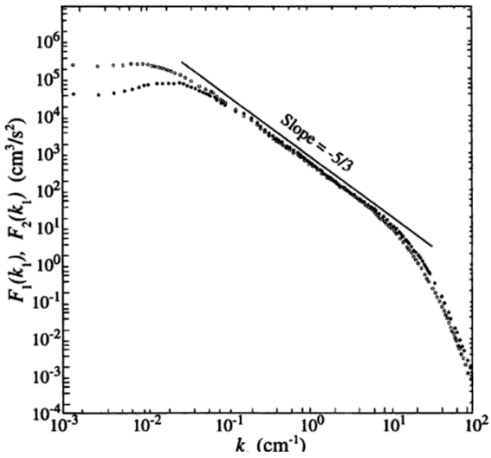


Figure 14.2 Experimental measurements of the energy spectra. The scaling law $E(k) \propto k^{\frac{-5}{3}}$ is clearly observed over many orders of magnitude. Remember, large wavenumbers correspond to small features. The rapid decline in energy for $k > 10$ cm^{-1} corresponds to where viscous dissipation takes over. From Champagne, *Journal of Fluid Mechanics*, 1978.

complexity that is observed in reality. However, the scaling law is robustly seen in experiments and thus contains some of the essential physics of the problem.

14.3 Numerical simulation of turbulence (CFD)

While CFD is a great tool and works robustly for low and "moderate" Reynolds number flows, at high Reynolds number flows become turbulent much more care is needed. One issue with CFD in turbulent flows comes from the picture of turbulence in the previous section and our example of the stirred bucker. Let's return to the 30 cm bucket at Reynolds number of 90,000. In order to resolve all the length scales with DNS we need computational grid that would require about $\mathrm{Re}^{3/4}$ or 6,000 discrete points in each spatial dimension. For a 3D flow this would be 2.16×10^{11} total grid points. This resolution is outside the range of current computing capability. So now let's say that we de-

cide that we only care about the big swirls so we decide to make our grid spacing 1 mm. Certainly this will suffice? Now we have 300 grid points in each direction; still a lot but not outside of what a modern computer and software could handle in 3D. However, 1 mm is 20 times greater than the length scale where viscosity can remove energy from the system. What happens is that energy will be put in at the large scale, that energy will cascade to smaller scales, and when the energy in the simulation reaches 1 mm (the grid spacing) it will just sit there. There is no mechanism to remove energy from the simulation. Energy will build up and the simulation will die a sad numerical death.

There are fixes to this problem by adding models which act as numerical sponges to soak up all the grid scale energy from the simulation. Methods called Large Eddy Simulations (LES) have this feature of being able to resolve our stirred bucket by modeling, but not resolving, the smallest scale motions. The LES models are often called turbulence closure models. There are many different models that have advantages and disadvantages in different situations. There is no single model that always works the best and it requires training and experience to do LES of turbulent flows well. You can easily see that this turbulence closure problem will be with us for a while. For an atmospheric flow which effect the climate, the Reynolds number can be 10^{15} the large scales structures are the sizes of continents. We are going to be continue to need closure models in weather and climate modeling in my lifetime.

Most modern CFD packages have a number of different turbulence closure models built in. The variety and character of these models is too extensive to cover here. In the software, it can be just as easy as clicking from one model to the next. However the different models have different limitations and cases where they work better or worse. Just be warned that you should be skeptical of the results unless you have had more training and experience than you are getting from this brief chapter.

Many people think that CFD is the answer to everything fluids. The impulse for many people when confronted with a fluids problem is to run to CFD. CFD can be powerful, but it is a tool that can be easily misused. If you are ever working somewhere and your boss wants to do CFD of a turbulent flow, be cautious going down this road unless you have learned more about fluid dynamics than this course. It is very easy to get a commercial code to "solve" but give you an incorrect answer.

You just need to be aware that CFD of turbulent, unstable, or otherwise non-linear flows is not yet plug and play and will probably remain this way for some time to come. CFD can be powerful, but you really need to know what you are doing. Like Stan Lee noted in Spiderman - CFD is powerful, "but with great power comes great responsibility".

15
Control volume analysis

In the previous chapters, we went through great effort to develop a differential law for fluid flow based on conservation of momentum for a material fluid particle. This conservation law holds at every point in space and along with appropriate boundary information allows (in principle at least) for the calculation of the velocity field at every point in space. We have calculated such velocity fields, at least for some simplified geometries. In many cases solving the velocity field is too complicated of a problem, or overkill. Maybe we only need to know the net force on an object and not the full velocity field. In many cases it is useful to turn to an integral form of the conservation laws for a control volume *fixed* in space. This control volume analysis is a macroscopic balance, the analysis will allow for us to solve for things such as the total force acting on an object in a flow due to the imbalance in the total fluid momentum going in and out of the volume. The control volume analysis does not allow us to calculate the details of the velocity field.

In order to get results out of a control volume analysis you usually need to be given some additional information (perhaps from a measurement) or you need to do some physical reasoning and make a some simplifying approximations. This fact is an important point to emphasize. You will often find that in order to make progress with a control volume problem you will have to assume something that may not seem obvious or you have difficulty justifying. For example, it will be common to assume that inlet flow fields are uniform in space with a single velocity. It will also be common to neglect viscous forces. In some cases these approximations will be great and in others, not so. What assumptions are reasonable come with experience. Don't be fooled by textbook

problems (including the ones I'll demonstrate) that tend to gloss over the assumptions that are made as though you just believe they are true. Like dimensional analysis, the *procedure* of control volume analysis is straightforward, it is the proper use of it that takes some physical insight that only comes with experience. The control volume is an excellent tool, but keep in mind that there are also problems where doesn't really help us much.

Control volume analysis is an excellent tool for getting order of magnitude estimates or simple scaling laws. Often in engineering situations we care about the approximate magnitude of the total force or the pressure, for example, and a simple control volume analysis can be more useful that a complex simulation. Often we can get a useful and reasonable answer with just a little effort. Control volume analysis is also a quick way to check other more detailed calculations or simulations. If you do a full analysis of the Navier-Stokes equations and solve for the full velocity field, but that field does not satisfy the overall control volume balance then you have done something wrong.

Many textbooks on fluid dynamics put the control volume analysis very early in the development. A good reason for doing so is that the analysis is much easier than analysis with the Navier-Stokes equations. However, when confronted with a real world problem that we don't find in a textbook executing a useful control volume analysis requires some physical intuition. Similar to dimensional analysis, control volume analysis is usually most powerful when coupled with good physical understanding and intuition about the important effects. Similar to dimensional analysis, it can be straightforward to learn the technique of control volume analysis, but using it wisely comes with experience.

15.1 Control volume formulation

Our general form of the conservation of momentum, integrated over an arbitrary fixed volume is,

$$\int \left(\rho \frac{D\mathbf{v}}{Dt} = \rho \mathbf{g} + \nabla \cdot \mathbf{T} \right) dV$$

We can now move the integral on the left hand side to a different form using the Reynolds Transport Theorem,

$$\int \rho \frac{D\mathbf{v}}{Dt} dV = \int \frac{\partial(\rho\mathbf{v})}{\partial t} dV + \int (\rho\mathbf{v})\mathbf{v} \cdot \mathbf{n} dS.$$

Using the divergence theorem we can also convert the stress tensor term on the right side of the original equation back to integration over a surface,

$$\int \frac{\partial(\rho\mathbf{v})}{\partial t} dV + \int (\rho\mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \int \rho\mathbf{g} dV + \int \mathbf{n} \cdot \mathbf{T} dS.$$

Substituting our constitutive law for a Newtonian fluid in incompressible flow ($\mathbf{T} = -P\mathbf{I} + 2\mu\mathbf{S}$), we have,

$$\int \frac{\partial(\rho\mathbf{v})}{\partial t} dV + \int (\rho\mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \int \rho\mathbf{g} dV - \int P\mathbf{n} dS + 2\mu \int \mathbf{n} \cdot \mathbf{S} dS.$$

Since we are now interested in a control volume fixed in space, the derivative on the left hand side can be moved outside the integral, so

$$\frac{d}{dt} \int \rho\mathbf{v} dV + \int (\rho\mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \int \rho\mathbf{g} dV - \int P\mathbf{n} dS + 2\mu \int \mathbf{n} \cdot \mathbf{S} dS. \quad (15.1)$$

The above expression holds for the fluid only. A more useful form includes an external force acting on an object in the flow. In this case, you can draw a control volume around whatever you like, as long as you remember to include the force which is holding the object in place.

$$\frac{d}{dt} \int \rho\mathbf{v} dV + \int (\rho\mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \int \rho\mathbf{g} dV - \int P\mathbf{n} dS + 2\mu \int \mathbf{n} \cdot \mathbf{S} dS + \mathbf{F}_{ext}. \quad (15.2)$$

Following the same ideas, the integral form of conservation of mass for a fixed control volume is

$$\frac{d}{dt} \int \rho dV + \int \rho\mathbf{v} \cdot \mathbf{n} dS = 0. \quad (15.3)$$

The integral form of the momentum and mass conservation equations for a fixed region can be very useful in a number of cases. The primary use is in making estimations/calculations of net forces acting on an object in a flow.

You should first note that conservation of momentum is a vector equation - 3 components for x, y and z. On the left side of Equation 15.1 we have the rate of change of momentum inside the control volume

(first term) and the net flux of momentum coming into/out of the control volume (2nd term). In the second term on the left side, note that the momentum $\rho\mathbf{v}$ is a vector quantity, while $\mathbf{v} \cdot \mathbf{n}$ is the rate that the momentum crosses the surface. On the right side, the first term is the net body force which acts on the whole volume. The second term is the force due to pressure and the third term is the force due to viscosity - both of these exert forces only on the surface of our volume. What is nice about this expression is there are many cases where we are only interested in the net force on an object, and in those cases we can often use the integral form of the equation without worrying about the details of the flow.

Getting the sign right of the term $\int(\rho\mathbf{v})\mathbf{v} \cdot \mathbf{n}dS$ can be a little tricky. You have be be careful that the velocity vector has a sign, as well as the term $\mathbf{v} \cdot \mathbf{n}$. Since we take normal vectors to point out of the volume, then $\mathbf{v} \cdot \mathbf{n}$ is positive for outflow and negative for inflow.

15.2 Examples

The only way we can make sense of the complex looking equation is to work through a few examples. The basic approach I usually take with these problems is as follows. I start by drawing a sketch and picking a control volume that seems convenient. A convenient control volume often (but not always) has surfaces where the velocity inlets and outlets are normal to the surface. I like to write out the entire control volume equation and then systematically cross out terms that I have good reason to ignore (say unsteady effects in a steady flow). I then would see where that takes me. If I can get an interesting result without making many assumptions, then great. If I don't get much then I might start making assumptions that I am less confident are true (say ignoring viscosity in a case where it is not clear I can). I like to try and keep track of my assumptions so when I get to the end of a problem, it is easy for me to review what I had to do to get there. One of the most common assumption you will tend to make in these problems is that the velocity field is uniform over some region, an assumption we often make because we have little choice.

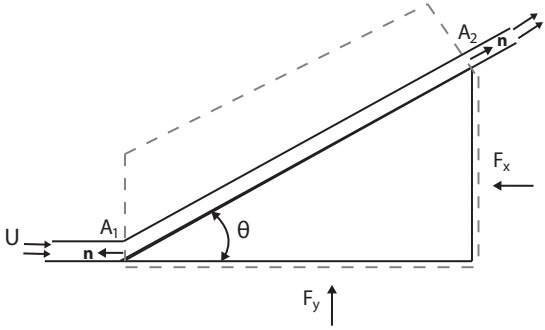


Figure 15.1 Control volume for finding the force a water jet exerts on an inclined block. The control volume is shown as the dashed line.

15.2.1 Example 1: Inclined block

Let's start with an hose spraying a block with a steady flow rate and the block is inclined with an angle θ and we are interested in calculating the total force exerted on the block. The mean velocity leaving the hose is known to be U and the area of the jet leaving the hose is A. We start by picking a control volume and experience will show you what volumes work best. In general, it works well to draw the volume such that it cuts through all fluid inlets and exits perpendicularly as shown in Figure 15.1. After drawing the volume it is useful to draw the normal vectors (pointing outward) where the fluid crosses the control volume boundary.

Now in order to make some progress we need to assume a few things. Let's start by assuming that since the flow is exiting the hose at a constant rate that we can neglect time dependent terms. Also, just for simplicity, lets neglect gravity which can be done by taking our apparatus to space, or by simply turning the device such that gravity points into the page in Figure 15.1. Let's start with conservation of mass for

$$\cancel{\frac{d}{dt}\int \rho dV} + \int \rho \mathbf{v} \cdot \mathbf{n} dS = 0.$$

We have two locations where we need to evaluate the surface integral, the inlet and outlet. We see that there is no way we can technically

evaluate the surface integral without knowing the velocity field of the inlet and outlet jet. Let's assume for now that the jet enters and exits with a uniform velocity field, U_1 and U_2 respectively. On the inlet the normal vector and the velocity are not aligned so the dot product is negative and on the exit, the velocity and the normal vector are aligned so the dot product is positive. Since the velocity at the inlet and exit is assumed constant, then the surface integral is easy to do,

$$\int \rho \mathbf{v} \cdot \mathbf{n} dS = -\rho U_1 A_1 + \rho U_2 A_2 = 0.$$

This statement simply says that the total mass flow rate in equals the total mass flow out. Note that when we write the surface integral above, it means that we have to evaluate $\rho \mathbf{v} \cdot \mathbf{n}$ at every point along the control volume surface and integrate it up. In this example it is only the two little patches where fluid comes in and goes out that has any fluid velocity.

Now let's consider the momentum equation. Let's start by writing out the full integral equation and cross out the unsteady terms and gravity which we previously stated we would neglect,

$$\cancel{\frac{d}{dt}\int \rho \mathbf{v} dV} + \int (\rho \mathbf{v}) \mathbf{v} \cdot \mathbf{n} dS = \cancel{\int \rho \mathbf{g} dV} - \int P \mathbf{n} dS + 2\mu \int \mathbf{n} \cdot \mathbf{S} dS + \mathbf{F}_{ext}.$$

Notice that the surface integral terms for pressure and viscosity are evaluated only on the surface of the control volume. So while viscous forces and pressure may do something inside the control volume on the surface we have drawn it would be reasonable to assume that they have no contribution. Since the control volume is open to atmosphere everywhere, the pressure around the surface of the control volume would be atmospheric. If we integrate a constant pressure around a closed loop, then the integral of $P\mathbf{n}$ around the loop results in zero net force. While viscosity certainly acts along the surface of the block, along the surface we have drawn the control volume boundary viscosity would not seem to play a role since we are assuming uniform flow outward. Recall that viscous stresses only occur when we have velocity gradients, so a uniform free jet with uniform flow has no viscous stresses. You are starting to see that in order for us to make progress we have to start to use some physical insight. By making these physical arguments to neglect the pressure and viscous terms on the control volume boundary,

we are left with

$$\int (\rho \mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \mathbf{F}_{ext}.$$

Since the momentum equation is a vector equation we have to consider each component, being careful with the signs. Let's start with the x-component of momentum. On the inlet at surface 1 the x component,

$$\int_1 (\rho \mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \int_1 (\rho U_1)(-U_1) dS = -\rho U_1^2 A_1.$$

Note that the dot product of $\mathbf{v} \cdot \mathbf{n} = -U_1$ since the velocity vector and normal vector are not aligned. Since the velocity at this surface is all in the x-direction, then $\mathbf{v} = U_1$. On the outlet at surface 2,

$$\int_2 (\rho \mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \int_1 (\rho U_2 \cos(\theta))(U_2) dS = \rho U_2^2 A_2 \cos(\theta).$$

Note that the dot product of $\mathbf{v} \cdot \mathbf{n} = U_2$ since the velocity vector and normal vector are aligned. Since the velocity at this surface is inclined at angle θ the x-component of the velocity is $U_2 \cos(\theta)$. Putting this together gives us the external force as,

$$\rho \left(U_2^2 A_2 \cos(\theta) - U_1^2 A_1 \right) = F_x.$$

Substituting in conservation of mass ($U_1 A_1 = U_2 A_2$) yields,

$$-\rho U_1 A_1 \left(U_1 - U_2 \cos(\theta) \right) = F_x.$$

So our final result still doesn't tell us everything we want since U_2 is not yet determined. We presumably know U_1 and A_1 as that is given by the size and flow rate of the hose. Without knowing U_2 or A_2 we have no way to get the force. If we measured these quantities we could proceed. We could also make an assumption that $A_1 \approx A_2$. While this seems like a reasonable thing to assume, it need not be true. If the area is the same then mass conservation says $U_1 = U_2$ and the x component of the force would be,

$$-\rho U_1^2 A_1 \left(1 - \cos(\theta) \right) = F_x.$$

In this case, the force is related to the deflection of x momentum out of the control volume. The rate that momentum is carried into the control volume is $\rho U_1^2 A_1$. The term $(1 - \cos(\theta))$ represents the fraction of x momentum that is deflected. If the angle were zero, a flat plate, then the force would be zero.

If we applied Bernoulli's equation along a streamline between points 1 and 2, in the absence of gravity we would have,

$$P_1 + \frac{1}{2}\rho U_1^2 = P_2 + \frac{1}{2}\rho U_2^2.$$

Since the pressure at points 1 and 2 would both be atmospheric, then Bernoulli's equation would tell us that $U_1 = U_2$. Bernoulli's equation is consistent with the assumption that the area of the jet is the same at the inlet and exit. Recall that Bernoulli's equation assumes inviscid flow, so our analysis with the assumption of equal area is consistent with the limit where viscosity is small compared to inertia - the high Reynolds's number limit. Our control volume result is consistent with the force on the block just due to changing the direction of inertia of the incoming flow.

Now let's consider y momentum. On the inlet at surface 1 the y component,

$$\int_1 (\rho \mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \int_1 (0)(-U_1) dS = 0.$$

As with the x component, the dot product of $\mathbf{v} \cdot \mathbf{n} = -U_1$ since the velocity vector and normal vector are not aligned. However, the velocity at this surface is all in the x-direction, then the y component of $\mathbf{v}$ is zero. On the outlet at surface 2,

$$\int_2 (\rho \mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \int_1 (\rho U_2 \sin(\theta))(U_2) dS = \rho U_2^2 A_2 \sin(\theta).$$

Note that the dot product of $\mathbf{v} \cdot \mathbf{n} = U_2$ since the velocity vector and normal vector are aligned. The dot product term is the same in the x and y momentum equations. Since the velocity at this surface is inclined the y-component is $\mathbf{v} = U_2 \sin(\theta)$. The total y component of the force is,

$$\rho U_2^2 A_2 \sin(\theta) = F_y.$$

When we make our assumption that the area of the jet does not change,

$$\rho U_1^2 A_1 \sin(\theta) = F_y.$$

Thus the final result for the case where the area of the jet does not change is

$$\mathbf{F}_{ext} = \rho U_1^2 A_1 \begin{bmatrix} \cos(\theta) - 1 \\ \sin(\theta) \end{bmatrix}$$

While this result assumes the constant velocity profile and the fact that the jet area remains the same from inlet to exit, it seems reasonable to think that scaling, order of magnitude and trends might be quite reasonable.

15.2.2 Example 2: Flat plate

Let's consider the same example but take the case where $\theta = 0$, a flat plate. We can use our results from the previous section for the force which can only have an x-component,

$$F_x = -\rho U_1 A_1 (U_1 - U_2)$$

If we assume that the area is something we can measure and put our result in terms of area, $U_2 = U_1 A_1 / A_2$,

$$F_x = -\rho U_1^2 A_1 \left(1 - \frac{A_1}{A_2}\right).$$

Given that it only make sense given the direction of the flow for the force to be in the negative x direction, this formula tells us that $A_1 < A_2$; the jet grows toward the exit. If the area is the same (as we assumed in the previous section) then the force would be zero.

Now let's rework the same problem with a different control volume that only includes the fluid. The same procedure would ensue as in the previous problem only we will need to retain the viscous term and since the plate is not in our control volume, there is no external force.

$$\cancel{\frac{d}{dt}\int \rho \mathbf{v} dV} + \int (\rho \mathbf{v})\mathbf{v}\cdot\mathbf{n} dS = \cancel{\int \rho \mathbf{g} dV} - \cancel{\int P\mathbf{n} dS} + 2\mu \int \mathbf{n}\cdot\mathbf{S} dS + \cancel{F_{ext}}.$$

Nothing would change in the evaluation of the left side of the equation with this new analysis, thus,

$$-\rho U_1^2 A_1 \left(1 - \frac{A_1}{A_2}\right) = 2\mu \int \mathbf{n}\cdot\mathbf{S} dS.$$

Note that the result from this other control volume says the total force on the plate is equal to the total viscous stresses on the plate. Since we do not deflect any momentum, the only x component of force is due to viscous drag. Any change in area of the jet would be related to the viscosity. Viscosity removes momentum from the fluid slowing it down and thus the area must grow accordingly to conserve mass. If this were

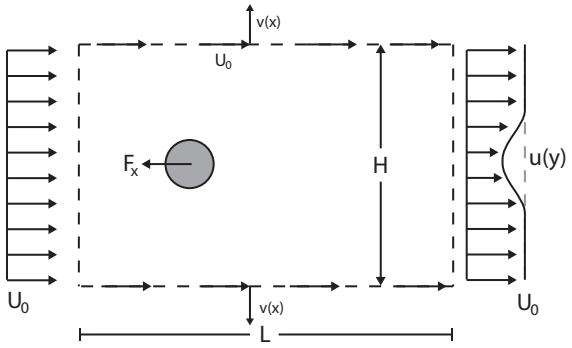


Figure 15.2 Control volume for finding the drag on an object from an experimentally measured wake.

a very viscous fluid coming from our hose, we would expect there to be a more dramatic change in the increased area of the jet. Using the control volume approach we can only compute the force on the plate if we had a measurement of the exit area.

If the fluid had very low viscosity (high Reynolds number), we could apply Benoulli's equation. If we applied Bernoulli's equation from the inlet to the exit, we would see that just like in the last example the velocity should not change. Since Bernoulli's equation is only valid for inviscid flow, our conclusion is consistent with the fact that at high Reynolds number, we expect the inlet and outlet areas of the jet to be approximately equal.

15.2.3 Example 3: Drag on an object

Let's see how we can compute the total drag on an object from a measurement of the surrounding fluid velocity field. The geometry and control volume are shown in Figure 15.2. We will consider four boundaries the left, right, top and bottom. For simplicity let's assume a symmetric object so the top and bottom should act the same. Let's consider steady flow. The flow coming in on the left is uniform with a known velocity U_0. The velocity on the right is measured as a function of y, $u(y)$.

Conservation of mass for steady flow in our control volume is,

$$\cancel{\frac{d}{dt}\int \rho dV} + \int \rho \mathbf{v} \cdot \mathbf{n} dS = 0.$$

Let's expand the surface integral into the four terms,

$$\int \rho \mathbf{v} \cdot \mathbf{n} dS = -\int_0^H \rho U_0 dy + \int_0^H \rho u(y) dy + 2\int_0^L \rho v(x) dx = 0.$$

The order of the terms are left, right, and top/bottom (hence the factor of 2). Note that on the inlet (left boundary) the velocity and normal vector point in opposite directions hence the negative sign on that term. Conservation of mass simply states that since there is a velocity deficit in the wake of the object, there must be some flow out the top and bottom surface.

Conservation of momentum is,

$$\cancel{\frac{d}{dt}\int \rho \mathbf{v} dV} + \int (\rho \mathbf{v}) \mathbf{v} \cdot \mathbf{n} dS = \cancel{\int \rho \mathbf{g} dV} - \cancel{\int P \mathbf{n} dS} + \cancel{2\mu \int \mathbf{n} \cdot \mathbf{S} dS} + F_{ext}.$$

We have crossed out terms since 1) the flow is steady, 2) we will ignore the effect of gravity (or assume perpendicular to the page), 3) the object is in an external flow field and thus the pressure is atmospheric everywhere, 4) that along the boundary of control volume which is far from the object there are no significant viscous stresses. Under our assumptions, the net force on the object is given by the imbalance in the momentum coming in and out of the control volume.

$$\int (\rho \mathbf{v}) \mathbf{v} \cdot \mathbf{n} dS = \mathbf{F}_{ext}.$$

Now, lets evaluate our terms for each surface for the x component of the momentum,

$$-\int_0^H \rho U_0^2 dy + \int_0^H \rho u(y)^2 dy + 2\int_0^L \rho U_0 v(x) dx = F_x.$$

The first term is the inlet. Since the flow and the normal vector are in opposite directions, $\mathbf{v} \cdot \mathbf{n} = -U_0$ and the x component of the velocity is U_0. The exit integral is computed similarly, only the sign is positive since the velocity and flow direction are aligned. The top and bottom terms should be treated with care. For the top and bottom, the outflow is $\mathbf{v} \cdot \mathbf{n} = v(x)$. The x component of velocity is U_0 since the surface is far from the object.

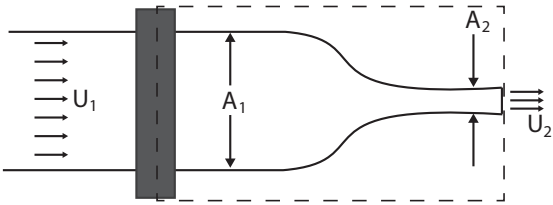


Figure 15.3 Control volume for finding the net force required to hold the nozzle on a pipe.

Substituting our conservation mass statement,

$$2\int_0^L \rho v(x)dx = \int_0^H \rho U_0 dy - \int_0^H \rho u(y)dy,$$

into the x momentum equation we get,

$$-\int_0^H \rho U_0^2 dy + \int_0^H \rho u(y)^2 dy + U_0\left(\int_0^H \rho U_0 dy - \int_0^H \rho u(y)dy\right) = \mathbf{F}_{ext}.$$

Canceling the offsetting terms we obtain,

$$-\rho\int_0^H u(y)\left(U_0 - u(y)\right)dy = F_x.$$

If we have a measurement of the velocity field in the wake of an object we can compute (or at least estimate) the total drag on the object.

15.2.4 Nozzle

Consider the tapered nozzle as shown in Figure 15.3. Lets consider a steady flow rate and we want to estimate the force required to hold the nozzle in place at the flange. We draw the control volume as shown. We will assume steady flow, ignore gravity, assume the inlet and outlet flow fields are uniform, and assume a high Reynolds number flow.

At this point, we can probably do the conservation of mass pretty easily,

$$\cancel{\frac{d}{dt}\int \rho dV} + \int \rho \mathbf{v}\cdot\mathbf{n}dS = -\rho U_1 A_1 + \rho U_2 A_2 = 0.$$

Conservation of momentum is,

$$\cancel{\frac{d}{dt}\int \rho \mathbf{v} dV} + \int (\rho \mathbf{v})\mathbf{v}\cdot\mathbf{n} dS = \cancel{\int \rho \mathbf{g} dV} - \int P\mathbf{n} dS + \cancel{2\mu \int \mathbf{n}\cdot\mathbf{S} dS} + \mathbf{F}_{ext}.$$

Here we have neglected the viscous terms, not because we believe that viscosity doesn't matter inside the nozzle but that viscosity probably doesn't add much to on the surfaces of the control volume that we have drawn. Since the flow is contained inside the nozzle, we must be careful to use different pressure. Now, evaluating the x component of this force and being careful with the signs,

$$-\rho U_1^2 A_1 + \rho U_2^2 A_2 = -(-P_1 A_1 + P_2 A_2 + P_\infty (A_1 - A_2)) + F_x.$$

In this example the momentum in and out of the control volume is similar to previous examples. What is new is how we handle the pressure. There are a few signs to keep track of. There is a negative in front of the pressure term. We also must account for the direction of the normal vector. Also, all of pressure at location 1 is acting on the area of the inlet, while the pressure at point 2 is acting only smaller exit area. The atmosphere is pushing on the right side of the control volume on the area $A_1 - A_2$. Since the pressure just outside the nozzle is in the open air, it is reasonable to set the pressure at the exit to be $P_2 \approx P_\infty$. In this case our equation becomes,

$$-\rho U_1^2 A_1 + \rho U_2^2 A_2 = (P_1 - P_\infty) A_1 + F_x.$$

Now we see in this form that the force to hold the nozzle is the difference in momentum coming in and out of the volume and the pressure at the nozzles inlet. We can use conservation of mass ($U_1 A_1 = U_2 A_2$)to write as

$$\rho U_1^2 A_1 \left(\frac{A_1}{A_2} - 1 \right) - (P_1 - P_\infty) A_1 = F_x.$$

Note the sign of the terms. The net change in momentum alone would lead to a force in the positive direction - meaning the nozzle is compressed on the tube. Since the flow shoots fast out of the nozzle there is a thrust like a rocket. The pressure term leads a negative force and wants to pop the nozzle off. We can't solve for the total force without more information.

One assumption we could make is assume that the flow is high

Reynolds number and apply Bernoulli's equation from the inlet to the outlet. Bernoulli would yield,

$$P_1 + \frac{1}{2}\rho U_1^2 = P_\infty + \frac{1}{2}\rho U_2^2$$

or

$$P_1 - P_\infty = \frac{1}{2}\rho U_2^2 - \frac{1}{2}\rho U_1^2.$$

Using conservation of mass gives,

$$P_1 - P_\infty = \frac{1}{2}\rho U_1^2 \left(\frac{A_1^2}{A_2^2} - 1\right)$$

Using the result from Bernoulli and going back to our momentum balance would give the nozzle force as

$$F_x = -\frac{1}{2}\rho U_1^2 A_1 \left(\frac{A_1}{A_2} - 1\right)^2$$

When $A_1 > A_2$ then the force acting on the nozzle is negative - meaning without a bolt holding the nozzle on, there is a tendency of the nozzle to pop off. The final result is only valid for high Reynolds number where Bernoulli's equation is reasonable.

15.3 Some difficulties

Sometimes, the control volume formulation does not lead us to a simple conclusion. Let's discuss a problem where the control volume approach does not give a very clear result. The problem seems straightforward and is known as the inverse sprinkler or Feynman's sprinkler. This problem is interesting because it seems that you should be able to get the answer by applying a control volume.It is called Feynman's sprinkler after the great 20th century physicist Richard Feynman who wrote of this problem in one of his books but never really provided what he thought his answer was.

A "normal" simple sprinkler is shown in Figure 15.4. It is free to pivot about the center point. If the entire sprinkler is submerged in a pool such that the sprinkler emits water into a water environment, most people would guess that the sprinkler spins clockwise. They would be right. Now if we consider the sprinkler to work in inverse mode, i.e.

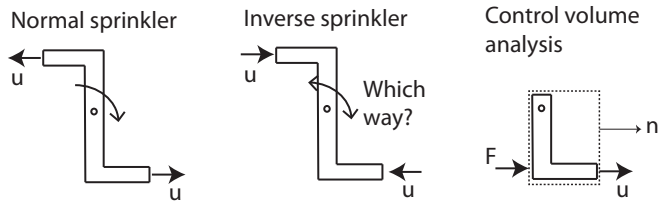


Figure 15.4 Schematic for the Feynman sprinkler problem. We show the normal sprinkler, which seems obvious to us should rotate clockwise. The direction of the inverse sprinkler is less clear. We also show the control volume used for the analysis.

water intakes into the sprinkler, the question is which way does the sprinkler spin? It seems the control volume analysis may be useful for this problem because we just want to know which way it spins, not how fast.

To analyze the problem, let's look at one half of the sprinkler, imagine that we hold the sprinkler fixed and calculate the force, F, needed to hold the normal sprinkler in place. Let's assume the water comes out as a jet of uniform velocity u and area A, the flow is steady, the density is constant, and viscosity is not important. Under these assumptions, we have,

$$\int (\rho \mathbf{v})\mathbf{v} \cdot \mathbf{n} dS = \int P\mathbf{n} dS + F$$

The only tricky part of control volume analysis is getting the sign correct and since we are trying to get the direction correct, we need to treat our signs with care. There are two signs to keep straight on the left side of the equation, $\mathbf{v} \cdot \mathbf{n}$ and $\mathbf{v}$. At the exit as drawn, both $\mathbf{v} \cdot \mathbf{n}$ and $\mathbf{v}$ are positive and thus we can estimate the term as $\int (\rho \mathbf{v})\mathbf{v} \cdot \mathbf{n} dS \approx \rho A u^2$. Normally, we would assume that the pressure at the exit is the same as the pressure in the environment. The force is positive and estimated to have a magnitude of $F = \rho A u^2$ and be positive as drawn in the schematic of figure 15.4. If we remove the force, the sprinkler spins.

Now consider the inverse sprinkler. The same integral expression must hold true. Momentum enters the control volume at the same rate as the sprinkler in forward motion. Since $\mathbf{v} \cdot \mathbf{n}$ and $\mathbf{v}$ are both negative, their product is positive; the momentum across the surface is the same in magnitude and sign as the sprinkler in normal operation. If this

were all, the sprinkler should spin the same direction whether the fluid comes in or out of the sprinkler. If one does the experiment (you can try yourself with a bendable drinking straw) you find that the sprinkler does not spin. If you did the experiment carefully you would actually find the sprinkler spins very slowly in the reverse direction.

The reason the control volume approach does not work is that we have not gotten the exit pressure right. In the sprinkler operating in "normal" mode the pressure at the exit must be at least a little higher at the exit than the environment since pressure gradients would be responsible for "pushing" the flow. When the sprinkler is operating in reverse mode, the pressure at the exit must be lower than the environment for flow to go in that direction. It turns out that if viscosity is not important then the low pressure at the sprinkler exit (which wants to make the sprinkler spin counterclockwise) exactly cancels the momentum coming in (which wants to make the sprinkler spin clockwise). I cannot derive the experimental result using control volumes. Control volume analysis gives you no way to calculate the pressure in this problem. While a number of people have written about this problem I have yet to find a solution that I find truly satisfying, nor have I been able to work one out myself.

16

Energy conservation in fluid flows

In the chapter on heat conduction we developed a conservation law for energy, but only considered energy flow by conduction in a stationary medium like a solid. We did not consider what happens if the medium is a moving fluid. When we introduced the idea of convection by a moving fluid we used mass transfer of dye molecules as our example case. Now we return to the idea of fluid motion carrying thermal energy around by convection. We waited to gain a little insight into fluid flow before tackling energy transport in convecting fluids.

Now that we have studied fluid mechanics a little bit, we can understand where the velocity field comes from and what it looks like in some cases. This puts us in a better position to convective heat transfer. We could have derived this law earlier in the book using the formalism we developed for conservation of mass and momentum, however it turns out that there are several subtle points in this derivation. To dig into the derivation we would need to understand a little more about thermodynamics than we have time to cover in a first introduction.

We are largely going to ignore some of the details and skip to the main result. Some details are provided in the appendix for the interested student and certainly even more detail can be found in a number of good references. In this chapter we will largely focus on the physical interpretation of the equations.

16.1 Energy equation

If we recall in the chapter on heat conduction in solids, the energy equation we derived was,

$$\rho c \frac{\partial T}{\partial t} = k \nabla^2 T.$$

The same equation was found when we discussed mass diffusion,

$$\frac{\partial c}{\partial t} = D \nabla^2 c.$$

If we now consider the media to be a moving fluid, we could perhaps guess from the chapter on mass transport that we would need to replace the time derivative with the material derivative,

$$\rho c \frac{DT}{Dt} = \rho c \left(\frac{\partial T}{\partial t} + \mathbf{v} \cdot \nabla T \right) = k \nabla^2 T.$$

It will turn out that this is the energy equation we will use in practice, however there are several subtleties that are buried in this equation. For now, we are going to sweep these subtleties under the rug.

To properly derive the equation we would need to dig into thermodynamics a little bit (which is done in Appendix B). The derivation also has several details, though it uses the same vector calculus machinery we developed to described conservation of mass and momentum. A little more detail of the derivation can be found in Appendix C for those who are interested. The main effect we would need to account for in our derivation of conservation of energy is that we need to consider the mechanical work done by the fluid flow. In the interest of time for the main text, let me skip the derivation and jump straight to the final result.

The resulting energy equation we use for incompressible flows is,

$$\rho c_p \frac{DT}{Dt} = k \nabla^2 T + 2\mu \mathbf{S} : \nabla \mathbf{v}. \tag{16.1}$$

From the derivation, two new features come out from the details. The first is that the specific heat is replaced more precisely with the constant pressure specific heat, c_p. This change does not matter for liquids where the constant pressure and constant volume specific heats are the same. The change is significant for gases. The second effect is a new term came out, which represent viscous heating. The operator ":" is a new

one for you. It is like a tensor dot product. You simply multiply each entry of the tensor by the respective entry of the other tensor and sum them all up. If the velocity field is known, in principle it is straight forward to compute the rate of viscous dissipation. Recall that $\mathbf{S}$ is the rate of strain tensor, $\mathbf{S} = \frac{1}{2}(\nabla\mathbf{v} + \nabla\mathbf{v}^T)$.

Physically, the second term on the right side of the equation is heat generated by viscous dissipation, or friction. Just like if you rub your hands together and they get warm, the friction internal to the fluid flow causes the fluid to heat up. It turns out that you can prove (via the Second Law of Thermodynamics) the last term always increases the thermal energy. If you shook up a bottle of fluid and let it come to rest by viscosity, the kinetic energy of the fluid in the bottle has to go somewhere, it goes into heating up the fluid.

The final equation reads as follows. The left side is the material derivative of the temperature of a fluid particle as it moves about. The temperature goes up or down based on the right hand side. The first term on the right side is the divergence of the heat flux. When we use Fourier's law for the heat flux, we obtain the classic diffusion term. This term acts just like in the conduction problem where it tends to smooth out any variations in the temperature field. The final term is the viscous heating; the fluid temperature can go up due to the internal friction losses in the flow.

16.2 Viscous heating - example calculations

Let's try computing the viscous heating for a few simple examples so we can get an idea of the order of magnitude and it will allow us to think about where the energy comes and goes in the fluid flow. Let's consider problems of steady fluid flow in a planar geometry bounded by two plates separated by distance H. We will consider flows such as Couette and Poiseuille flow where the flow is constant in the axial direction and there are no gradients along the channel. If you recall, when we assume steady flow that is uniform in x, conservation of mass states that there is no vertical velocity. The axial velocity is a function of y only; $u(y)$.

Let's write out the the 2D energy equation in component form,

$$\rho c_p \left(\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} \right) = k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) + 2\mu \mathbf{S} : \nabla \mathbf{v}$$

Under these assumptions our equation simplifies because the convection terms are removed from the equation

$$\rho c_p \left(\frac{\partial T}{\partial t} + \cancel{u\frac{\partial T}{\partial x}} + \cancel{v\frac{\partial T}{\partial y}} \right) = k \left(\cancel{\frac{\partial^2 T}{\partial x^2}} + \frac{\partial^2 T}{\partial y^2} \right) + 2\mu \mathbf{S} : \nabla \mathbf{v}$$

giving,

$$\rho c_p \frac{\partial T}{\partial t} = k\frac{\partial^2 T}{\partial y^2} + 2\mu \mathbf{S} : \nabla \mathbf{v}.$$

The 2D strain rate tensor in general is,

$$\mathbf{S} = \frac{1}{2} \begin{bmatrix} 2\frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & 2\frac{\partial v}{\partial y} \end{bmatrix},$$

but in our simple flow where $v = 0$ and $u(y)$ this tensor reduces to,

$$\mathbf{S} = \frac{1}{2} \begin{bmatrix} 0 & \frac{\partial u}{\partial y} \\ \frac{\partial u}{\partial y} & 0 \end{bmatrix}.$$

Thus the viscous heating term is

$$2\mu \mathbf{S} : \nabla \mathbf{v} = \mu \left(\frac{\partial u}{\partial y} \right)^2$$

We are left with the simplified equation for 2D plane parallel flows,

$$\rho c_p \frac{\partial T}{\partial t} = k\frac{\partial^2 T}{\partial y^2} + \mu \left(\frac{\partial u}{\partial y} \right)^2. \tag{16.2}$$

Couette flow - Insulated walls

For the case of Couette flow the velocity field is linear, $\partial u/\partial y = U/H$ and the viscous heating term is

$$\mu \left(\frac{\partial u}{\partial y} \right)^2 = \mu \frac{U^2}{H^2}.$$

To solve equation 16.2 we need to know the thermal boundary condition. If we take the walls to be thermal insulators, there is no temperature gradient at the wall, $\partial T/\partial y = 0$. The heating term is constant

throughout the domain, so we expect the temperature of the fluid to be constant throughout the volume. Thus, we can integrate the entire energy equation across the domain from $y = 0$ to $y = H$,

$$\rho c_p H \frac{\partial T}{\partial t} = k \int_0^H \frac{\partial^2 T}{\partial y^2} dy + \mu H \frac{U^2}{H^2}$$

or

$$\rho c_p \frac{\partial T}{\partial t} = \frac{k}{H} \left(\left. \frac{\partial T}{\partial y} \right|_{y=H} - \left. \frac{\partial T}{\partial y} \right|_{y=0} \right) + \mu \frac{U^2}{H^2}$$

and using the insulated boundary condition where the temperature gradient is zero at the walls,

$$\rho c_p \frac{dT}{dt} = \mu \frac{U^2}{H^2}.$$

The equation states that fluid temperature increases at a constant rate,

$$\frac{dT}{dt} = \frac{\nu}{c_p} \frac{U^2}{H^2}.$$

The mechanical power delivered to the fluid goes into heating the fluid up. For water with $\nu = 10^{-6}$ m^2/s and $c_p = 4186$ J/kgK, and if we assumed the upper plate was moving at 1 m/s, you would have to have a geometry with a 1 mm gap to achieve 1 degree Celsius heating per hour. A 1 mm gap for a plate moving at 1 m/s is pretty fast. You typically have to have pretty extreme conditions to obtain high rates of viscous heating.

We could also have derived this result in a simple way. The stress the fluid exerts on the upper plate is proportional to the velocity gradient, $\tau = \mu U/H$. The force on the plate is the viscous stress multiplied by the area of the plate, $F = A\mu U/H$. The mechanical power that the plate is giving to the fluid is the force multiplied by velocity, i.e. Power $= A\mu U^2/H$. The power per unit volume is,

$$\frac{\text{Power}}{AH} = \frac{\mu U^2}{H^2},$$

where AH is the volume of fluid between the two plates. The quantity $\mu U^2/H^2$ is the number of Watts per unit volume which must be taken by the fluid and converted into thermal energy when the system is insulated. This result is equivalent to the viscous heating term above.

Couette flow - Constant temperature walls

We can also integrate Equation 16.2 for constant temperature boundary conditions in Couette flow. We would expect if the walls were held at constant temperature then the system would reach a steady state in time where the mechanical power put into the fluid balanced the amount of heat flowing out. At steady state, the $\partial/\partial t$ term in the equation would be zero. Therefore we would need to solve,

$$k\frac{\partial^2 T}{\partial y^2} = -\mu\frac{U^2}{H^2}$$

with conditions that $T(y=0) = T(y=H) = T_{\text{wall}}$. Since the right side of the above equation is constant, we can easily integrate to obtain,

$$T(y) = -\frac{\mu U^2}{2kH^2}y^2 + C_1 y + C_2.$$

Using the boundary conditions to obtain the two constants of integration we have,

$$T(y) = \frac{\mu U^2}{2kH^2}y(H-y) + T_{\text{wall}}.$$

The temperature difference relative to the walls in the center of the channel is,

$$T\left(y = \frac{H}{2}\right) = \frac{\mu U^2}{4k}.$$

For water, flow speeds would need to reach about 50 m/s to obtain 1 degree C of temperature difference between the fluid and the wall. Similarly to the previous example, it usually takes fairly "extreme" conditions to get significant viscous heating. Note that in this case the temperature rise does not depend upon the gap size.

We could have arrived at this result using dimensional analysis. However, it is probably not obvious to a new student that the parameters, U, μ, and k are the only ones that matter. While after the fact it may seem "obvious" to state that the fluid density and specific heat don't matter, it is less clear from the start that this should be true.

Poiseuille flow

Let's consider as a final example, our problem of 2D pressure driven Poiseuille flow between two plates. For the thermal part of the problem, let's consider the case of the walls held at constant temperature

T_{wall}. In this example we will consider the steady state temperature distribution in the fluid. Equation 16.2 is still valid in this situation. Let's try something different and simply integrate the above expression across the domain

$$\int_0^H \left(\rho c_p \frac{\partial T}{\partial t} + k \frac{\partial^2 T}{\partial y^2} = -\mu \left(\frac{\partial u}{\partial y} \right)^2 \right) dy.$$

which becomes,

$$\frac{d}{dt} \left(\int_0^H \rho c_p T dy \right) + k \left(\left. \frac{\partial T}{\partial y} \right|_H - \left. \frac{\partial T}{\partial y} \right|_0 \right) = -\mu \int_0^H \left(\frac{\partial u}{\partial y} \right)^2 dy.$$

The first term on the left side of the equation is the time rate of change of the thermal energy stored (per unit length of channel) in the bulk of the fluid. The second term on the left side of the equation is the heat flux leaving the fluid through the upper and lower walls. If we multiply the equation by the channel length, the left side becomes the total energy stored in the fluid plus the total heat lost over the length of channel. So let's rewrite the equations in words,

$$\frac{d}{dt} \text{Energy stored} \; + \; \text{total heat loss} = \mu L \int_0^H \left(\frac{\partial u}{\partial y} \right)^2 dy$$

Now the right side can be transformed by using a trick you may have forgotten; integration by parts. Don't groan. Integration by parts is awesome. Just check out what happens,

$$\frac{d}{dt} \text{Energy stored} \; + \; \text{total heat loss} = \mu L \left(u \frac{\partial u}{\partial y} \right) \Bigg|_0^H - \mu \int_0^H u \frac{\partial^2 u}{\partial y^2} dy.$$

The first term on the right side is zero because of the no-slip condition on the velocity at the walls. Therefore we are left with,

$$\frac{d}{dt} \text{Energy stored} \; + \; \text{total heat loss} = -\mu L \int_0^H u \frac{\partial^2 u}{\partial y^2} dy.$$

Recalling that in this problem conservation of momentum states that $\Delta P = \mu L \frac{\partial^2 u}{\partial y^2}$. The second derivative of the velocity is a constant, making the integral a bit easier,

$$\frac{d}{dt} \text{Energy stored} \; + \; \text{total heat loss} = \int_0^H u \Delta P dy = \Delta P \; Q,$$

recalling the total volumetric flow rate was calculated as, $Q = \int_0^H u dy$.

In fluid mechanics the pressure difference times volumetric flow rate has units of power, energy per unit time. It's nice that in this simple problem we can obtain a nice intuitive result. The mechanical power (measured in Watts) added to the fluid is equal to the number of Watts of heat loss that leaves the fluid plus the rate you add heat to the fluid. We have essentially gone back to a fundamental statement of the first law of thermodynamics

$$\frac{d}{dt}\text{Energy stored} = \text{Mechanical work in } - \text{ total heat out.}$$

At steady state where the temperature of the fluid is a constant in time, then the mechanical power in equals the heat flow out. Of course since our original equations were stated in terms of these kind of overall energy balances, it is not surprising that we obtain the result we did. The only point of this section on Poiseuille flow was to attempt to connect to our ideas of energy, and make connections between the mathematical formulations that we have been discussing in this chapter.

16.3 Final equations

Above we computed the amount of viscous heating in a Couette flow. While in a more general geometry we cannot easily compute the viscous heating, we can estimate the size of the term $S : \nabla \mathbf{v}$ as $\mu U^2/L^2$ where U is the typical velocity and L is an estimate of the length scale over which velocity gradients occur. Taking the ratio $\mu U^2/(\rho c_p L^2)$ gives a simple estimate to how quickly the fluid is heating due to viscous heating. Typically, this rate will be so slow we can ignore it. Do you feel the pool water getting hotter when you go for a swim?

Note that we simply are able to discard the viscous heating term in the equation. Earlier, we saw trouble when we simply tried to remove viscosity from the momentum equation to yield Euler's equations. The problem with Euler's equation was that when we removed the small term we changed the order of the highest derivative and fundamentally changed the nature of the equation. In the current case with viscous dissipation the nature and order of the equation are unchanged. The boundary conditions you would apply are the same in this case. It is mathematically fine to just discard the term.

While it is convenient to discard the viscous heating term in calculations, conceptually it is important to retain the term in your mind at the very least. Without the viscous heating term in the equation, energy is not conserved. Set a fluid in motion and it will come to rest. That energy has to go somewhere, and it goes into heating up the fluid. If you discard the viscous heating term, you discard this critical piece of fundamental physics. However, in practice if the heating is small then the error in neglecting the term is of no practical consequence.

Under this further restriction of neglecting viscous dissipation and assuming constant conductivity, the energy equation we will use is in practice is,

$$\rho c_p \frac{DT}{Dt} = k\nabla^2 T. \tag{16.3}$$

This equation is the one we put forth at the beginning of the chapter, with the exception that we have been careful about which specific heat to use. We should also note at this time that the above equation is identical that put forth in the chapter on mass transfer. Heat and mass transport share many of the same mathematical solutions and physical pictures.

The restrictions on the above equation may not be fully clear at this point given the looseness of the derivation. A more rigorous and detailed treatment requires us to delve in little deeper into thermodynamics, which I want to avoid at this time. I suggest if you ever use these notes as a reference after this course for problems involving the energy equation that you consult some of the references for more detail so that you are sure you know what you are doing. Equation 16.3, has a lot of subtleties and you should be careful in its use.

16.4 Coupled flow and heat transfer

For cases where we wish to compute heat transfer under the influence of fluid flow, in principle we would need to solve the coupled system of

three equations for the velocity, pressure, and temperature,

$$\begin{aligned} \nabla \cdot \mathbf{v} &= 0 \\ \rho \frac{D\mathbf{v}}{Dt} &= -\nabla P + \rho \mathbf{g} + \mu \nabla^2 \mathbf{v} \\ \rho c_p \frac{DT}{Dt} &= k \nabla^2 T. \end{aligned}$$

This is a well-posed coupled equation set for conservation of mass, momentum, and energy. With appropriate boundary and initial conditions, the equations can be solved and then we know everything about the velocity and temperature fields for every point in time and space. However, we have already learned that solving fluid flow problems on their own are very complicated especially when flows are turbulent.

We should take care to note the coupling between these equations. First, the coupling of the momentum equation into the thermal energy equation is direct. The velocity field convects thermal energy (through the D/Dt operator), thus the flow determined by the Navier-Stokes directly feeds into the thermal energy equation by carrying energy around the system.

The primary coupling of the temperature equation on the velocity field is that the fluid density changes with temperature. These density changes can give rise to fluid flow through a buoyancy force, an effect called natural convection. The thermal equation feeds back into the momentum equation through the gravitational body force. Any hawk can attest that natural convection is quite important in the atmosphere with hot air at the earth's surface rising in thermal updrafts. When hot food is placed in the counter, it is natural convection which carries away most of the heat to cool your pizza. It is easy to underestimate the importance of natural convection. When you are just sitting in a room, your body temperature gives rise to fairly vigorous flow around you even though our senses for flow don't really notice.

Most fluids becomes less dense with increases in temperature resulting in hot fluid rising. A notable exception is water right around its freezing point. Over much of the temperature range, water becomes more dense as it gets colder. Thus in a lake or ocean, the cold water sinks to the bottom - a fact you can confirm by diving deep in a lake on a hot summer day. However, as water reaches the freezing point, the density begins to decrease as it gets colder. Thus the water which is

just above freezing floats back to the top and then freezes as a sheet of ice on the lake surface. If water did not have this property, then lakes and oceans would freeze from the bottom up, which would really be unfortunate.

It is important to note viscosity and thermal conductivity can be a strong functions of temperature, however such effects are beyond the scope of our course. In the equations above we have assumed constant transport properties and written the equations as such. Finally, while we often don't worry about the viscous heating term in practical calculations, it is conceptually important that you don't forget about it. In the above formulation without the viscous heating, a homogenous box of fluid which has initial kinetic energy will come to rest and all the initial energy will be gone. The energy can't be found. Conservation of energy is violated. Of course, the truth is that the fluid has heated a little and the mechanical energy went into thermal form. However, the scales are such that the temperature rise is very often (but not always) small enough that we can ignore this effect in practice.

17
Convective heat transfer

In the previous chapter (and associated appendices) we derived the equation for conservation of energy in a fluid flow. In principle the coupled equations for conservation of mass, momentum, and energy can be solved (most often by a computer). We have already learned that solving fluid flow problems on their own are complicated and coupling in the transport of heat makes things even more challenging. In many applications, the Reynolds number is high and thus analytical and even numerical simulations may not be feasible. The problem of convective heat transfer seems even more difficult than the fluid motion that has already troubled us so much.

In this chapter we will use the equations to guide a dimensionless formulation which provide some insight. This formulation will provide two key dimensionless numbers for convective heat transfer - one being the Reynolds number that we have met so many times. The other dimensionless parameter is the Prandtl number which we met briefly in our early work on dimensional analysis. Since analytical solutions are complex, we will often resort to an empirical correlation expressed in our dimensionless parameters. These empirical correlations can then be readily used to make real predictions. This approach may seem an easy way out. We are developing all these mathematical equations and then we are going to give up and go to a handbook? Yes. However, this approach is very powerful as we will provide a simple way to get real engineering numbers.

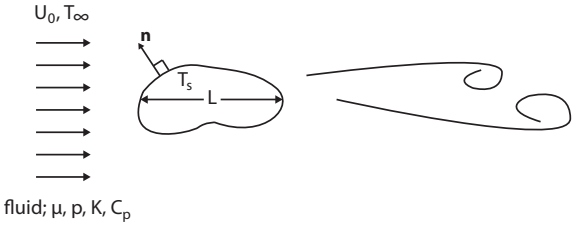


Figure 17.1 Schematic for the forced convection problem with external flow.

17.1 Forced convection

In this section we will consider the heat transfer from a solid object immersed in a fluid flow. We will assume the fluid density is constant and that the fluid velocity is forced by external means, as would be the case for a fan blowing over an object. The setup is shown in Figure 17.1. Since the fluid is constant density, the effect of gravity on the flow is neglected. For this case we need the coupled system of equations for the velocity, pressure, and temperature,

$$\begin{aligned} \nabla \cdot \mathbf{v} &= 0 \\ \rho \frac{D\mathbf{v}}{Dt} &= -\nabla P + \mu \nabla^2 \mathbf{v} \\ \rho c_p \frac{DT}{Dt} &= k \nabla^2 T. \end{aligned}$$

The coupling in this case is only one way. You need to solve the mass and momentum equation first for the velocity field. This now known velocity field would then be used to solve the temperature field. In order to solve the equations we need boundary conditions. For the velocity field we use no-slip at the solid surface and for temperature let's consider that the object is held at a fixed constant temperature which is hotter than the surrounding fluid.

We can make these equations dimensionless as we have done before, considering a constant density flow. Let's say the geometry of the object provides some characteristic length, ℓ, and some characteristic flow velocity, U_0. Let's define $\tilde{\mathbf{v}} = \mathbf{v}/U_0$, $[\tilde{x}, \tilde{y}, \tilde{z}] = [x, y, z]/\ell$, $\tilde{t} = tU_0/\ell$, and $\tilde{P} = P/P_0$ where we set $P_0 = \rho U_0^2$. Making a simple substitution and

change of variables we obtain,

$$\left(\frac{\partial \tilde{\mathbf{v}}}{\partial \tilde{t}} + \tilde{\mathbf{v}} \cdot \tilde{\nabla}\tilde{\mathbf{v}}\right) = \tilde{\nabla}\tilde{P} + \frac{1}{\mathrm{Re}}\tilde{\nabla}^2\tilde{\mathbf{v}}, \tag{17.1}$$

$$\tilde{\nabla} \cdot \tilde{\mathbf{v}} = 0. \tag{17.2}$$

This is precisely the same equation we got previously when we only considered fluid flow. Of course this result should not be surprising since we are assuming no coupling from the thermal problem to the fluid flow problem. Without solving the equations we can state that the dimensionless velocity field depends upon the Reynolds number,

$$\tilde{\mathbf{v}}(\tilde{t}, \tilde{\mathbf{x}}, \mathrm{Re}).$$

We are interested in finding the heat transfer from a solid surface of temperature T_s to a fluid at a temperature of T_∞. We can define our dimensionless temperature as $\tilde{T} = (T - T_\infty)/(T_s - T_\infty)$. This definition will have a fluid temperature of 0 far from the hot object, and a temperature of 1 at the solid surface. If you go back to our analysis of conduction, this is the same scaling we assumed in those problems. Applying the change of variables gives,

$$\frac{D\tilde{T}}{D\tilde{t}} = \frac{1}{\mathrm{RePr}}\tilde{\nabla}^2\tilde{T}, \tag{17.3}$$

where the Prandtl number,

$$\mathrm{Pr} = \frac{\nu}{\alpha},$$

is defined as the ratio of the diffusivity of momentum to the diffusivity of heat. A fluid like an oil which is very viscous (momentum diffuses easily) but is not a good conductor of heat has a high Prandtl number. A fluid like a liquid metal has a low Prandtl number since it is a good conductor of heat but can flow easily. The Prandtl number of air is about Pr=0.7 and for water Pr=7. Making the equations dimensionless says that for a given geometry the temperature field will depend on two dimensionless parameters;

$$\tilde{T}(\tilde{t}, \tilde{\mathbf{x}}, \mathrm{Re}, \mathrm{Pr}).$$

Now we wish to compute the rate that heat flows from the solid

object into the fluid. In dimensional terms, the local heat flux out of the surface is,

$$q = \mathbf{n} \cdot k \nabla T,$$

where the temperature gradient is the temperature gradient of the fluid at the surface and k is the conductivity of the fluid. In terms of the dimensionless temperature,

$$q = \frac{k}{\ell}(T_s - T_\infty)\mathbf{n} \cdot \nabla \tilde{T}$$

Recall that earlier, we defined the convection coefficient, h, as $q = h(T_s - T_\infty)$. Thus rearranging yields,

$$\mathrm{Nu} = \frac{h\ell}{k} = \mathbf{n} \cdot \nabla \tilde{T}$$

The dimensionless grouping $h\ell/k$ is called the Nusselt number, Nu. The Nusselt number is the dimensionless heat transfer coefficient. The Nusselt number is useful in that it is a function of only the Reynolds number, the Prandtl number, and position along the solid object's surface,

$$\mathrm{Nu} = f(\tilde{t}, \tilde{\mathbf{x}}, \mathrm{Re}, \mathrm{Pr}).$$

While this functional form can be complicated, we have a vast simplification from the original dimensional problem.

Above we stated that in general the Nusselt number is a local quantity that can change in time due to natural fluctuations in the flow and by location around the object. In many cases we will be interested only in a total amount of heat flow from an object, thus we would only need the averaged heat transfer coefficient across the surface. The total amount of heat flowing from the object to the fluid comes from integrating heat flux around the surface and averaging in time,

$$q = \int \mathbf{n} \cdot k \nabla T dS.$$

The average heat transfer coefficient, would then be defined through,

$$q = \bar{h} A (T_s - T_\infty),$$

where A is the surface area and T_s is the average surface temperature. We define the average Nusselt number through this averaged heat

transfer coefficient,

$$\overline{\mathrm{Nu}} = \frac{\bar{h}\ell}{k}.$$

The average dimensionless heat transfer coefficient, $\overline{\mathrm{Nu}}$ is a function of only two parameters,

$$\overline{\mathrm{Nu}} = f(\mathrm{Re}, \mathrm{Pr}).$$

We are back to the same result that we had in Chapter 3 that we arrived at by using dimensional analysis.

17.1.1 Example: Cylinder at constant temperature

Let's work through an example of the hot cylinder immersed in a cross flow with a cold fluid. We have a few options. One is that we could try analysis, however we would find it fairly intractable. If the Reynolds number were low, we could actually solve the velocity field and thus we might have a chance. However, for even moderate Reynolds number (i.e. $\mathrm{Re} > 10$) we can't solve for the velocity field, let alone the full convection formulation. If we can't do analysis we could resort to simulation which is feasible for larger Reynolds numbers, though care (and better knowledge of simulation methods) must be taken once we exceed $\mathrm{Re} > 1000$. If we don't want to do simulation we can use experiments and create and empirical relationship for the Nusselt number. This empirical approach is what we introduced in Chapter 3.

For common geometries, one can find a number of different correlations for the Nusselt number for flow over a cylinder. We will use the correlation of the form

$$\overline{\mathrm{Nu}} = C\mathrm{Re}^m \mathrm{Pr}^{\frac{1}{3}}.$$

The details of this correlation can be found in the heat transfer references such as (Incroprera and DeWitt (2001)). In their book the provided values for C and m depend upon the Reynolds number. The ranges for the experimentally determined constants are for $4 < \mathrm{Re} < 40$, $C = 0.911$, $m = 0.385$; $40 < \mathrm{Re} < 4,000$, $C = 0.683$, $m = 0.466$; $4000 < \mathrm{Re} < 40,000$, $C = 0.193$, $m = 0.618$.

Let's use concrete example and take a cylinder held at 350K in air flow at 300K. The flow velocity will be 0.1 m/s and the diameter of the sphere is 2 cm. To evaluate the fluid properties, I will use the average

temperature of 325 K to evaluate the properties. Note that this choice (as opposed to just using ambient conditions) will change the exact answer we get, but the impact is not very dramatic. We first compute the Reynolds number,

$$\mathrm{Re} = \frac{\rho U D}{\mu} = \frac{1.07\left(\frac{\mathrm{kg}}{\mathrm{m}^3}\right)\quad 0.1\left(\frac{m}{s}\right)\quad 0.02\,(m)}{0.0000196\left(\frac{\mathrm{kg}}{\mathrm{ms}}\right)} \approx 109.$$

The Prandtl number for air is $Pr = 0.7$.

Since the Reynolds number is around 100 we use the appropriate values in the empirical formula for the Nusselt number

$$\overline{\mathrm{Nu}} = 0.683 \times 109^{0.466} \times 0.7^{\frac{1}{3}} = 5.4$$

Therefore to find the average heat transfer coefficient

$$\overline{\mathrm{Nu}} = \frac{\overline{h} D}{k} = 5.4$$

$$\overline{h} = \frac{5.4 \times 0.0282\ \frac{\mathrm{W}}{\mathrm{mK}}}{0.02\ \mathrm{m}} = 7.6\ \frac{\mathrm{W}}{\mathrm{m}^2\mathrm{K}}$$

Given the circumference of the cylinder, the total heat transfer is

$$Q = 7.6\ \frac{\mathrm{W}}{\mathrm{m}^2}\ (\pi 0.02\ \mathrm{m})\,(350\ \mathrm{K} - 300\ \mathrm{K}) = 24\ \frac{\mathrm{W}}{\mathrm{m}}$$

Note the simplicity of the methodology here. Once we have an empirical formula for the Nusselt number - which generating the first time is the real hard work - it is just a matter of computing the appropriate dimensionless numbers.

We can also solve the problem computationally using commercial simulation software, shown in Figure 17.2. The simulation solves the fully coupled flow and heat transfer problem. Given the moderate Reynolds number around 100, the simulation is able to generate a solution. If we increase the Reynolds number significantly above 1000, then we would need to start using some special tricks to achieve a solution at these higher Reynolds numbers. Remarkably, the numerical solution provide the total heat leaving the cylinder as 24 W/m; in near perfect agreement with the correlation. While good agreement is not surprising, it is probably a bit of luck that the case I selected here had perfect agreement. If I had found agreement of 25 % I would have considered this a perfect match.

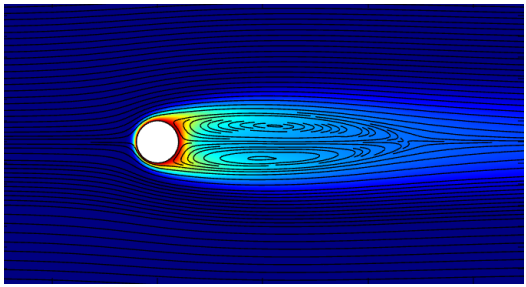


Figure 17.2 Numerical solution to the convective problem of air flow around a cylinder. Light regions are hot and the fluid is cold. Contour lines are streamlines of the flow.

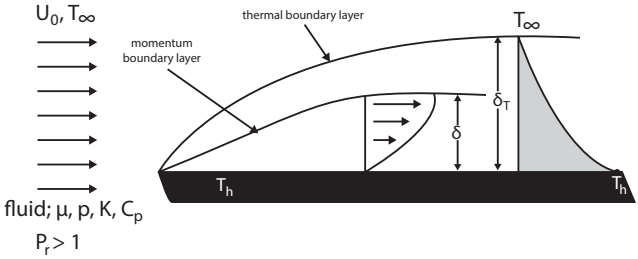


Figure 17.3 Schematic for the thermal boundary layer problem. The thickness of the boundary layers are exaggerated.

17.2 Flat plate thermal boundary layer

The laminar boundary layer over a flat plate at constant temperature is an important case where we can get some insight into the interplay of diffusion and convection of heat. We have a cool fluid approaching a hot flat plate and now we wish to see how temperature (in addition to momentum) diffuses from the flat plate is convected downstream as shown in Figure 17.3. How quickly does the thermal boundary layer develop? Let's start by using some simple physical arguments to see if we can guess the result first. The physical picture is the same as when we discussed the momentum boundary layer. Let's think about how we expect the size of the thermal boundary layer to develop as we move down the plate. When there is no fluid motion, heat diffuses a distance of the thermal boundary layer size δ_t over some time t according to the

dimensional scaling,

$$\delta_t \sim \sqrt{\alpha t}.$$

This scaling is the same as momentum diffusing due to the kinematic viscosity and is what we discussed in the conduction chapters earlier in this book. In the boundary layer problem, the time for heat to diffuse in a steady flow is given by the time it takes a fluid particle to convect some distance down the plate; $t \sim x/U_\infty$. Therefore we might expect the thickness of the thermal boundary layer to be

$$\delta_t \sim \sqrt{\frac{\alpha x}{U_\infty}}.$$

Recasting this in dimensionless form we obtain,

$$\frac{\delta_t}{x} \sim \sqrt{\frac{\alpha}{xU_\infty}} \sim \sqrt{\frac{1}{\mathrm{Re}_x \mathrm{Pr}}}.$$

where $\mathrm{Re}_x = Ux/\nu$ is the local Reynolds number. Recall from previous discussion that the momentum boundary layer that $\delta \sim \sqrt{1/\mathrm{Re}_x}$. Thus our argument would suggest that the ratio of the momentum to thermal boundary layer scales as,

$$\frac{\delta}{\delta_t} \sim \sqrt{\mathrm{Pr}}.$$

This result indicates an important idea. When the Prandtl number is 1, then the thickness of the thermal boundary layer and the momentum boundary layer are the same. Air has a Prandtl number of about 0.7, so the two boundary layers are of comparable size. When the Prandtl number is large (as in an oil), momentum diffuses quicker than heat. In this case the thermal boundary layer is much thinner than the momentum boundary layer. In the other extreme of low Prandtl number (a liquid metal) heat diffuses more readily than momentum and the thermal boundary layer is relatively thick compared to the momentum one.

Using similar scaling arguments we might be tempted to argue that the heat flux is dominated by conduction across the thermal boundary layer. Using this scaling to estimate the local heat transfer coefficient would yield,

$$h(x) \sim \frac{k}{\delta_t} \equiv k\sqrt{\frac{U}{\alpha x}}$$

and thus the Nusselt number could be re-written as,

$$\mathrm{Nu}_x = \frac{h(x)k}{x} \sim \mathrm{Re}_x^{\frac{1}{2}} \mathrm{Pr}^{\frac{1}{2}}.$$

While this result seems quite plausible, it turns out to be not quite correct.

Following the same type of analysis that was discussed for the momentum boundary layer, we can actually solve the boundary layer problem analytically. I will not present the details and just give the final result, however it comes from a solution of our coupled equations of conservation of mass, momentum, and energy. This analytical solution is only good for laminar flow where $\mathrm{Re} < 10^5$. Even though we can obtain an analytical result, the result is not a simple function and we an expression must be evaluated numerically. This analytical expression is fit with a simple power law. For $0.6 < \mathrm{Pr} < 50$ it is found that the local Nusselt number is well-described by (Incroprera and DeWitt (2001)),

$$\mathrm{Nu}_x = \frac{hx}{k} = 0.332 \ \mathrm{Re}_x^{\frac{1}{2}} \mathrm{Pr}^{\frac{1}{3}}.$$

From the solution we obtain the ratio of two boundary layers as,

$$\frac{\delta}{\delta_t} \sim \mathrm{Pr}^{\frac{1}{3}}.$$

Note that even though we have an analytical expression, it is not easily evaluated so we resort to a simpler empirical description.

When we tried to derive this result using dimensional arguments, we got the proper scaling with Reynolds number but not the proper result for Prandtl number. Our dimensional arguments provided a one-half power with Prandtl number rather than the real result which is a one-third power. The difference lies in that our simple scaling analysis did not account for convection *inside* the boundary layer. The while the fluid flow is mostly in the downstream direction, conservation of mass demands that there is a small fluid flow across the boundary layer (see the previous discussion on boundary layers). This fluid flow while small, is also acting over a small distance and thus the vertical convection in the boundary layer cannot be neglected. This convection plays a role in setting the properties of the thermal boundary layer size and resulting heat transfer. While our simple dimensional analysis indicates the right qualitative behavior with Prandtl number, the result is not correct. We need to full analysis to get the power law correct.

From the analysis which gave the local Nusselt number, the local heat transfer coefficient is given as,

$$h = 0.332\ k \left(\frac{\rho U}{x\mu}\right)^{\frac{1}{2}} \mathrm{Pr}^{\frac{1}{3}} \quad \text{[Laminar]}.$$

Note that the local heat transfer coefficient decreases as we progress down the plate since the boundary layer thickness grows. The heat transfer coefficient is also infinite at the leading edge where $x = 0$, though of course this is not really physical. The average heat transfer coefficient is defined as

$$\overline{h} = \frac{1}{\ell}\int_0^{\ell} h(x)dx.$$

Performing the calculation yields the average Nusselt number,

$$\overline{\mathrm{Nu}} = \frac{\overline{h}\ell}{k} = 0.664\ \mathrm{Re}_{\ell}^{\frac{1}{2}}\mathrm{Pr}^{\frac{1}{3}} \quad \text{[Laminar]}$$

where the Reynolds number is based on the plate length.

When the flow is turbulent ($\mathrm{Re} > 10^5$), a commonly used and experimentally determined correlation for the local Nusselt number (Incroprera and DeWitt (2001)) when $0.6 < \mathrm{Pr} < 60$, is

$$\mathrm{Nu} = \frac{hx}{k} = 0.0296\mathrm{Re}_x^{\frac{4}{5}}\mathrm{Pr}^{\frac{1}{3}}. \quad \text{[Turbulent]}$$

Again, this expression is not a fundamental law in any sense, it is just a function that fits the data well.

For a plate where the flow is turbulent some care must be taken when evaluating the average Nusselt number. The first part of the plate is laminar then the flow transitions sometime after the local Reynolds number exceeds 10^5. To get the combined average heat transfer, we integrate the local expression over the laminar and turbulent regions and add them up. In this case the average heat transfer coefficient would be calculated as,

$$\overline{h} = \frac{1}{\ell}\left(\int_0^{x_c} h_{lam}dx + \int_{x_c}^{\ell} h_{tur}dx\right),$$

where x_c is the location where the critical Reynolds number is reached. If we include the laminar regime and assume a transition Reynolds

number of $\mathrm{Re} = 10^5$, then performing the integral would yield,

$$\overline{\mathrm{Nu}} = \frac{h\ell}{k} = \left(0.037\mathrm{Re}_\ell^{\frac{4}{5}} - 160\right)\mathrm{Pr}^{\frac{1}{3}} \quad \text{[Turbulent]}.$$

If the flow transitions to turbulence very early on the plate, then one could just ignore the laminar region with little error.

17.3 Natural convection

A subtle change to the equations and physics can have a dramatic impact. In some cases, the fluid flow is not forced but occurs naturally due to buoyancy. The density of a fluid typically decreases with increasing temperature and thus warm fluid will tend to rise. As warm fluid rises from a hot object, thermal energy is convected away. This natural convection is how a hot pizza sitting on the counter-top cools. The change to the equations is that the density is now temperature dependent (again let's ignore the problem of viscosity and thermal conductivity varying with temperature). With a variable density, it now becomes unclear whether we can assume the velocity field has zero divergence or whether we need to resort to the complete statement of conservation of mass. To deal with all this properly we would need to really dig deeper into thermodynamics which we are not going to do here.

Since this whole thing is getting complicated, we very often make an approximation known as the Boussinesq approximation which is a great (yet accurate) simplification. It is assumed that the change in density from the nominal value, ρ_∞, is small relative the total density; this is an assumption that needs to be checked but is good in many cases. The usual approximation is that the density of the fluid can be written as $\rho = \rho_\infty(1+\beta(T-T_\infty))$, where β is the volumetric expansion coefficient. It is a property of the material that you can look up. For a gas $\beta = \frac{1}{T}$ where T is the temperature in Kelvin. If the change in density due to heating is small, it is safe to assume that the variable density only matters in the gravitational body force. Further, it is assumed that the variation of the properties with temperature is negligible. With all

these assumptions, the equations become,

$$\begin{aligned}\nabla \cdot \mathbf{v} &= 0\\ \rho_\infty \frac{D\mathbf{v}}{Dt} &= -\nabla(P + \rho_\infty g z) + \rho_\infty \beta (T - T_\infty)\mathbf{g} + \mu \nabla^2 \mathbf{v}\\ \rho_\infty c_p \frac{DT}{Dt} &= k\nabla^2 T.\end{aligned}$$

Note that I am just providing the final result and intentionally not deriving these equations in detail. If you need more detail, a derivation is easily found in a number of references. The important thing is that now there is a force in the momentum equation which is proportional to the temperature. This is the buoyant force that will cause less dense air to rise. The coupling between the equation is now two way, *the velocity field convects the fluid temperature and the temperature drives the flow.*

Making the equations dimensionless just as in the previous section yields,

$$\nabla \cdot \tilde{\mathbf{v}} = 0 \tag{17.4}$$

$$\frac{D\tilde{\mathbf{v}}}{D\tilde{t}} = -\nabla \tilde{P}_d + \frac{\beta \mathbf{g}(T_s - T_\infty)\ell}{U_0^2}\tilde{T} + \frac{1}{\mathrm{Re}}\nabla^2 \tilde{\mathbf{v}} \tag{17.5}$$

$$\frac{D\tilde{T}}{D\tilde{t}} = \frac{1}{\mathrm{RePr}}\nabla^2 \tilde{T}. \tag{17.6}$$

Since we are considering problems where the fluid velocity is driven by the buoyancy effects, there is no natural choice for U_0 as in the forced convection problem, thus we are free to select $U_0 = \sqrt{\beta \mathbf{g}(T_s - T_\infty)\ell}$. This choice would set the final momentum equation to be

$$\frac{D\tilde{\mathbf{v}}}{D\tilde{t}} = -\nabla \tilde{P}_d + \tilde{T} + \frac{1}{\mathrm{Re}}\nabla^2 \tilde{\mathbf{v}} \tag{17.7}$$

where $\mathrm{Re} = \sqrt{\mathrm{Gr}}$ where in problems with natural convection it is common to define the Grashof number as,

$$\mathrm{Gr} = \frac{\beta \mathbf{g}(T_s - T_\infty)\ell}{U_0^2}\left(\frac{\rho_\infty U_0 \ell}{\mu}\right)^2 = \frac{\beta \mathbf{g}(T_s - T_\infty)\ell^3}{\nu^2}.$$

The Grashof number is the ratio of the buoyant force to the viscous force. While we can think of the Grashof number as appearing like the Reynolds number in the equation we give it a new name by tradition because the definition of the Grashof number has no driven velocity field as part of the definition.

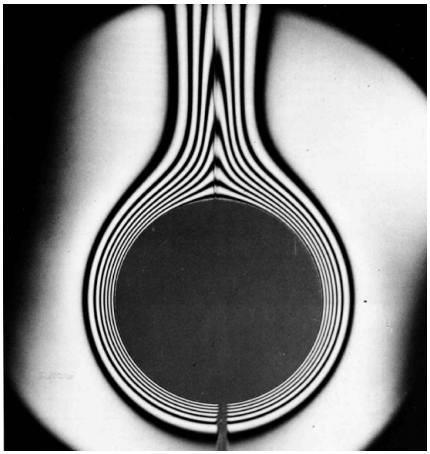


Figure 17.4 Experimental image of natural convection from a hot cylinder in a cool fluid. From Album of Fluid Motion.

If we repeated the analysis of computing heat transfer from an object due to natural convection we would find that the average Nusselt number is now a function of the dimensionless numbers;

$$\overline{\mathrm{Nu}} = f(\mathrm{Gr}, \mathrm{Pr}).$$

In practice, it turns out to be useful to define the Rayleigh number as

$$\mathrm{Ra} \equiv \mathrm{GrPr}.$$

You will see the Rayleigh number appear in a number of correlations for Nusselt number.

As a rule of thumb, the average dimensional natural convection coefficient in air is usually about 10 $\mathrm{W/m^2K}$. By about I mean, maybe it might really be 5 or 20 for your application. The point is that it is not usually 100 or 1. I find keeping this order of magnitude of 10 $\mathrm{W/m^2K}$ in my head very useful.

17.3.1 Example: Cylinder

Consider the case of a warm cylinder of diameter D in a cold room. The air around the cylinder is still other than the flow driven by natural convection. We know from our analysis that the average Nusselt number

can only be a function of the Grashof (or Rayleigh) number and the Prandtl number,

$$\overline{\mathrm{Nu}} = f(\mathrm{Gr}, \mathrm{Pr}) = f(\mathrm{Ra}, \mathrm{Pr}).$$

One empirical correlation for the cylinder heat transfer is written as

$$\overline{\mathrm{Nu}} = C\mathrm{Ra}^n$$

where different values of C and n are provided for a different range of Rayleigh number (Incroprera and DeWitt (2001)).

Let's work through one numerical example, using the same cylinder as in the forced convection problem, a 2 cm diameter cylinder held at 350 K immersed in air at 300 K. We first need to compute the Rayleigh number,

$$\mathrm{Ra} = \frac{g\beta(T_s - T_\infty)D^3}{\nu\alpha}$$

$$= \frac{9.8\ \left(\frac{\mathrm{m}}{\mathrm{s}^2}\right)\ \frac{1}{325}\ \left(\frac{1}{\mathrm{K}}\right)\ 50\ (\mathrm{K})0.02^3\ (\mathrm{m}^3)}{1.84 \times 10^{-5}\left(\frac{\mathrm{m}^2}{\mathrm{s}}\right)\ 2.62 \times 10^{-5}\left(\frac{\mathrm{m}^2}{\mathrm{s}}\right)} \approx 25,000$$

We then look at the textbook (Incroprera and DeWitt (2001)) and see that for this range of Rayleigh number that we should use $C = 0.48$ and $n = .25$. Using these values gives

$$\overline{\mathrm{Nu}} = 0.48\mathrm{Ra}^{0.25} = 6$$

Finally, we find the heat transfer coefficient by,

$$\overline{\mathrm{Nu}} = 6 = \frac{\overline{h}D}{k}$$

$$\overline{h} = \frac{6 \times 0.0282\ \frac{\mathrm{W}}{\mathrm{mK}}}{0.02\ \mathrm{m}} = 8.5\ \frac{\mathrm{W}}{\mathrm{m}^2\mathrm{K}}$$

I early claimed that $h \approx 10\ \mathrm{W/m^2K}$ for natural convection in air, and at least this one case has vindicated my claim.

17.4 Vertical flat plate boundary layer

The vertical flat plate is the classic boundary layer problem for natural convection. Here the fluid flow is driven by buoyancy only. In the laminar regime there is an analytical solution that is quite similar to the

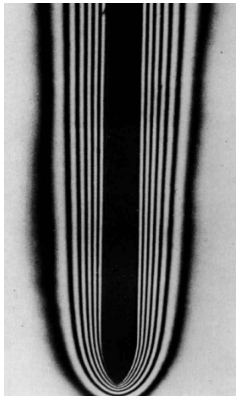


Figure 17.5 Experimental image of natural convection from a hot vertical flat plate in a cool fluid. From Album of Fluid Motion.

one that is found for the forced thermal boundary layer. Again, it is not a simple answer and must be evaluated numerically. The solution for the laminar flow case, the average Nusselt number is

$$\mathrm{Nu} = \left(\frac{\mathrm{Gr}_x}{4}\right)^{\frac{1}{4}} \frac{0.75\mathrm{Pr}^{\frac{1}{2}}}{(0.609 + 1.221\mathrm{Pr}^{\frac{1}{2}} + 1.238\mathrm{Pr})^{\frac{1}{4}}}$$

The part of this expression with the Grashof number to the 1/4 power is from the analysis while the weird equation with the Prandtl number is a fit to the numerically calculated solution. Again it is interesting that we can get a analytical expression but that it is difficult to evaluate so we resort to empirical fits to analytical results. The average Nusselt number is found by integrating with respect to x,

$$\overline{Nu} = \frac{4}{3}\mathrm{Nu}_x$$

For the vertical plate in natural convection the experimentally observed transition to turbulence occurs around

$$\mathrm{Ra} < 10^9 \text{for laminar flow.}$$

When the Rayleigh number is too high, then we expect the flow to be turbulent. With turbulent flow, as always, we cannot calculate anything

Figure 17.6 Experimental image of Rayleigh-Benard convection. In this experiment the lower plate is heated and the upper plat is held cool. From Album of Fluid Motion.

directly. Thus one empirical fit to the data has the form,

$$\overline{\mathrm{Nu}} = \left(0.825 + \frac{0.387 \mathrm{Ra}^{\frac{1}{6}}}{\left(1 + (0.492/\mathrm{Pr})^{\frac{9}{16}}\right)^{\frac{8}{27}}} \right)^2$$

Don't read too much into the powers of 8/27 and the constants. This is just an empirical expression that describes the data with reasonable accuracy.

17.5 Rayleigh-Benard convection

A very important and interesting problem in natural convection is known as Rayleigh-Benard convection. Here the problem is that we have two parallel plates separated by a small distance with a fluid in between. The bounding plates are held at fixed temperature with the bottom plate hotter than the upper one. Gravity points downward. When the temperature difference is small, heat flows from hot to cold by conduction. The fluid near the lower hot plate is less dense and wants to rise. However the geometry of the solid plates constrains the flow since for some fluid to move up, some other fluid must move down. If the temperature difference is small then the conduction state is stable and the fluid stays at rest. If the temperature difference is increased, then at some point the situation becomes unstable and convection rolls can set in, seen in Figure 17.6. Some fluid rises and other fluid sinks. The critical parameter is the Rayleigh number and convection sets in when $Ra > 1708$. In this case the critical Reyleigh number can be determined through analysis and the result matches experiments quite well. The critical parameter is computed via a linear stability analy-

sis, meaning one takes the simple conduction state with no flow and assumes small linear perturbations from that state.

This is a well studied problem with a lot of interesting experimental behavior. Interesting large scale patterns can emerge with one seeing regimes with stripes, hexagons, nearly perfect spirals, spirals with defects, or completely chaotic flow. The system is one of the most widely studied pattern forming systems where beautiful large scale regular patterns form spontaneously. I highly recommend a brief search on the images associated with this problem to get a sense for the richness associated with such a simple physical system.

17.6 Natural and forced convection

Finally some problems have a forced and natural convection component. If we have a very hot object with a weak fan blowing on it, it seems that we could have both forced and natural convection effects. We need to be more quantitative about whether a problem is dominated by forced convection, natural convection, or a little of both.

Let's consider the same problem as in natural convection only we will add a forced flow across the object with a velocity U_0. The dimensionless formulation from before is,

$$\nabla \cdot \tilde{\mathbf{v}} = 0 \tag{17.8}$$

$$\frac{D\tilde{\mathbf{v}}}{D\tilde{t}} = -\nabla \tilde{P}_d + \frac{\beta \mathbf{g}(T_s - T_\infty)\ell}{U_0^2}\tilde{T} + \frac{1}{\mathrm{Re}}\nabla^2 \tilde{\mathbf{v}} \tag{17.9}$$

$$\frac{D\tilde{T}}{D\tilde{t}} = \frac{1}{\mathrm{RePr}}\nabla^2 \tilde{T}. \tag{17.10}$$

Let's take the scale for U_0 to be the driven velocity for forced convection. In this case we would have a dimensionless formulation of,

$$\nabla \cdot \tilde{\mathbf{v}} = 0 \tag{17.11}$$

$$\frac{D\tilde{\mathbf{v}}}{D\tilde{t}} = -\nabla \tilde{P}_d + \frac{\mathrm{Gr}}{\mathrm{Re}^2}\tilde{T} + \frac{1}{\mathrm{Re}}\nabla^2 \tilde{\mathbf{v}} \tag{17.12}$$

$$\frac{D\tilde{T}}{D\tilde{t}} = \frac{1}{\mathrm{RePr}}\nabla^2 \tilde{T}. \tag{17.13}$$

In this case the Reynolds number is defined by the usual way $\mathrm{Re} = \rho U_0 \ell/\mu$

Our dimensional analysis let's us quickly estimate which form of convection is more important or if both matter. For a given mixed problem we can compute a Reynolds number based on the driven velocity field and a Grashof number. We can then compute the number, $\mathrm{Gr}/\mathrm{Re}^2$, and decide whether forced or free convection is dominant. If $\mathrm{Gr}/\mathrm{Re}^2$ is a small number, free convection can be neglected. We are back to the equations used for forced convection. If $\mathrm{Gr}/\mathrm{Re}^2$ is a big number it means that natural convection dominates over any externally imposed flow field. If $\mathrm{Gr}/\mathrm{Re}^2 \approx 1$ then both forced and natural convection will play a role in the problem and we would find the proper dimensionless expression would be,

$$\overline{\mathrm{Nu}} = f(\mathrm{Re}, \mathrm{Gr}, \mathrm{Pr}).$$

We will not consider any examples here where both natural and forced convection matter.

17.7 Simulation

The same simulation tools we discussed which are available for computing fluid flow can also be used for convection calculations. Generally the more details we add to a problem the more complex the flows and numerical solutions become. For forced convection with no gravitational effects, if the Reynolds number is not too large and the geometry is not too complex, then there is a good chance that we can get a good numerical result without investing in too much expertise using modern simulation software. A problem such as the 2D laminar boundary layer or flow around a cylinder at moderate Reynolds number, can be easily computed. Once the velocity and temperature fields are computed, it is straightforward to extract the average convection coefficient. As with flow problems, the higher the Reynolds number the more complex the numerical solution will be and the more care, compute time, and expertise is required. I am being purposely vague in what I mean by "high" Reynolds number since what is consider high continually increases as computing power increases. Many of the details of what is consider a "high" Reynolds number depends upon the problem at hand.

Natural convection calculations are also quite amenable to solution with modern simulation software. However, with increased problem

complexity comes the increasing need to really understand the physics of the problem at hand and know what we are doing. The current state of the available software is that it is much more difficult for a novice to generate good solutions to natural convection problems. While it requires a little more care, such solutions are certainly possible with a little physical insight and experience with the particular software. It was not that long ago that solving natural convection problems with even simple geometries was a very challenging undertaking and one that was the topic of a PhD thesis. Today, we can solve these problems readily, though perhaps it is still a bit challenging for a novice user in a first undergraduate course of fluid mechanics and heat transfer. With time, the bar for what is considered a challenging numerical problem will continue to be lowered. In terms of simulation the ideas in this chapter are powerful, one set of dimensionless simulations can be used to explain a wide range of physical experiments.

Appendix A
Solution to the 1D transient heat equation

The partial differential equation that governs diffusion processes is one that we can solve analytically in some simple cases. Here we will consider one-dimensional transient conduction.

Take the standard (dimensionless) heat equation defined on a domain $0 < x < 1$

$$\frac{\partial T}{\partial t} = \frac{\partial^2 T}{\partial x^2}, \tag{A.1}$$

and assume that the solution is composed of the product of two functions one that is a function of x and the other is a function of t.

$$T(x,t) = \Theta(t)X(x). \tag{A.2}$$

This technique is called separation of variables. I have never found a satisfactory reason as to why we should know to guess this form of the solution a priori. However, this technique works on a lot of problems. We can prove existence and uniqueness for the heat equation. Therefore, if we find a solution that satisfies the equation and the boundary conditions, then we have found the solution regardless of the way we got there.

Substitution of the assumed form into the heat equation becomes

$$\frac{d\Theta(t)}{dt}X(x) = \Theta(t)\frac{d^2X(x)}{dx^2}, \tag{A.3}$$

which is rewritten as

$$\frac{1}{\Theta(t)}\frac{d\Theta(t)}{dt} = \frac{1}{X(x)}\frac{d^2X(x)}{dx^2}. \tag{A.4}$$

Since everything on the left side of the equation is a function of time

only, and everything on the right side of the equation is a function of space only, the only possible solution is that both sides of the equation equal a constant;

$$\frac{1}{\Theta(t)}\frac{d\Theta(t)}{dt} = \frac{1}{X(x)}\frac{d^2X(x)}{dx^2} = -\lambda^2. \tag{A.5}$$

The minus sign and the power of two are put in the equation arbitrarily. This will not effect the answer but they will make the answer come out a little nicer in the end. The reason for the negative sign will be apparent shortly. Since I know the answer ahead of time I will include these factors now. We can now solve each of the problems independently;

$$\frac{1}{\Theta(t)}\frac{d\Theta(t)}{dt} = -\lambda^2. \tag{A.6}$$

which is easily rearranged

$$\frac{d\Theta(t)}{\Theta} = -\lambda^2 dt, \tag{A.7}$$

and integrated to yield

$$\log(\Theta) = -\lambda^2 t + C, \tag{A.8}$$

where C is a constant of integration. The above is equivalent to,

$$\Theta = e^{-\lambda^2 t + C} = e^C e^{-\lambda^2 t} \tag{A.9}$$

and since C is an arbitrary constant

$$\Theta = Ce^{-\lambda^2 t}. \tag{A.10}$$

It is clear why the negative sign was needed when choosing the constant. If the constant was positive, the solution would grow out of control with positive feedback. The negative sign allows a nice gentle decay as we would expect from physical reasoning.

Now we turn to the equation that was a function of x;

$$\frac{1}{X(x)}\frac{d^2X(x)}{dx^2} = -\lambda^2. \tag{A.11}$$

We have seen this equation several times and can integrate it directly or simply go straight to the general solution,

$$X(x) = A\sin(\lambda x) + B\cos(\lambda x) \tag{A.12}$$

where A and B are constants of integration. You may now see why we chose λ^2 as our constant. Combining our expressions for X and Θ we obtain,

$$T(x,t) = e^{-\lambda^2 t}(A\sin(\lambda x) + B\cos(\lambda x)) \tag{A.13}$$

Note that the constant of integration C is absorbed with A and B. You can verify that this expression satisfies the governing equations by simple substitution.

A.1 Applying boundary conditions

There are three unknowns, A, B, and λ. To find these constants we need to apply initial and boundary conditions to the equation. We will solve the problem where T is initially uniform and at $t > 0$ the boundaries are held fixed at zero for all times. These conditions are called the quenching problem. Different boundary/initial conditions would follow the same procedure. The conditions for the quenching problem, can be written as;

$$T(x=0,t>0)=0; \qquad T(x=1,t>0)=0; \qquad T(0<x<1,t=0)=1 \tag{A.14}$$

When $x = 0$,

$$T(x=0,t) = e^{-\lambda^2 t}B = 0, \tag{A.15}$$

can only be satisfied when $B = 0$. When $x = 1$,

$$T(x=0,t) = e^{-\lambda^2 t}A\sin(\lambda) = 0. \tag{A.16}$$

The only way this equation can be satisfied is that $A = 0$ or when $\lambda = n\pi$ where n is an integer; $n = 0, 1, 2, ...$ The second option is the only interesting solution. Since the solution is linear, any combination of solutions corresponding to any acceptable value of n is a solution to the governing equation and satisfies the boundary conditions.

At this point our solution looks like,

$$T(x,t) = \sum_{n=0}^{\infty} A_n e^{-(n\pi)^2 t}\sin(n\pi x). \tag{A.17}$$

To get our constants A_n, we need to apply the initial condition,

$$T(x, t = 0) = \sum_{n=0}^{\infty} A_n \sin(n\pi x) = 1. \tag{A.18}$$

We now need a way to compute the A_n so that the above expression holds true. This method will seem odd at first, however, just bear through the solution and convince yourself that it works. To find the coefficients A_n you can use the following integral properties. These properties can be derived or you can look them up in a calculus textbook, but we will just state the result here:

$$\int_0^1 \sin(n\pi x)\sin(m\pi x)dx = 0; \qquad \text{n and m are integers, n} \neq \text{m} \tag{A.19}$$

$$\int_0^1 \sin(n\pi x)\sin(n\pi x)dx = 1/2; \qquad \text{n is an integer} \tag{A.20}$$

Using these properties of integrals allows us to extract the coefficients. We multiply A.18 by $\sin(m\pi x)$ and integrate the function across the domain;

$$\int_0^1 \sum_{n=0}^{\infty} A_n \sin(n\pi x)\sin(m\pi x) = \int_0^1 \sin(m\pi x) \tag{A.21}$$

The sum results in many terms, however they are all zero except for the one where $n = m$. Using the fact that most terms in the sum are zero under this operation, we are left with,

$$\int_0^1 A_n \sin(n\pi x)\sin(n\pi x) = \int_0^1 \sin(n\pi x). \tag{A.22}$$

Both sides of this expression can be integrated analytically such that

$$A_n = \frac{4}{n\pi} \qquad \text{when} \quad n = 1, 3, 5, 7 \tag{A.23}$$

and

$$A_n = 0 \qquad \text{when} \quad n = 0, 2, 4, \tag{A.24}$$

The final solution can be written as an infinite sum,

$$T(x, t) = \sum_{n=1,3,5,..}^{\infty} \frac{4}{n\pi} e^{-(n\pi)^2 t} \sin(n\pi x). \tag{A.25}$$

While the expression is difficult to visualize, it is easy to plot. The

sum is taken to infinity, however in practice it need not be. The smoother the function, the fewer terms are needed to represent the solution as a sum of sines. The convenient aspect of this solution is that we can find the temperature profile at any instant in time without finding the solution at any earlier times as we must do when we numerically integrate.

A.2 How are we finding A_n?

Consider the sample problem we just worked with the same boundary condition, but take the initial condition is an arbitrary function, $f(x)$, we would need to solve the following equation

$$T(x, t = 0) = \sum_{n=0}^{\infty} A_n \sin(n\pi x) = f(x) \tag{A.26}$$

The technique we used to extract the coefficients A_n certainly seems strange the first time through; however, it is not at all strange as we will now show through analogy with a simpler example.

An ordinary vector, F, has components in three dimensional space; x,y, and z. The vector can be represented with a basis set of the three units vectors, $\hat{\mathbf{i}}$, $\hat{\mathbf{j}}$ and $\hat{\mathbf{k}}$, pointing in three dimensions. Each of the units vectors are perpendicular (or orthogonal) to each other. To describe the vector F, each unit vector has some magnitude, A, that represents the projection (shadow) of the F in that direction.

$$A_x\hat{\mathbf{i}} + A_y\hat{\mathbf{j}} + A_z\hat{\mathbf{k}} = \mathbf{F}. \tag{A.27}$$

If F is a known and I asked you to find the coefficient A_x, you would tell me that it is trivial. To find any of the A coefficients, we simply take the inner (or dot) product of the whole expression with each of the unit vectors in turn. When F is known, we find A_x by

$$(A_x\hat{\mathbf{i}} + A_y\hat{\mathbf{j}} + A_z\hat{\mathbf{k}} = \mathbf{F}) \cdot \hat{\mathbf{i}}. \tag{A.28}$$

Since the inner product of any two of the different unit vectors is zero, and the inner product of like unit vectors is one we obtain

$$A_x = \mathbf{F} \cdot \hat{\mathbf{i}}. \tag{A.29}$$

This example may seem rather trivial and obvious given that it is so familiar. The procedure is so simple since the original basis vector set

had the nice property that the inner product of any two unlike units vectors is zero.

Now consider our functional expression

$$A_1\sin(\pi x)+A_2\sin(2\pi x)+A_3\sin(3\pi x)+A_4\sin(4\pi x)+...=f(x) \quad \text{(A.30)}$$

and let us find the coefficients in the exact same way. Think of each function $\sin(n\pi x)$ as a basis "vector". In this function space the operation, $\int_0^1 f(x)\sin(n2\pi x)dx$ will serve as the inner product of our "vector" f with the n^{th} basis "vector" $\sin(n\pi x)$. The inner product operation satisfies the same conditions as our usual operation with three dimensional basis vectors. We get zero for the inner product of any combination of basis functions, except when we take the inner product with itself. For example,

$$\int_0^1 \sin(\pi x)\sin(2\pi x)dx = 0; \quad \text{(A.31)}$$

$$\int_0^1 \sin(\pi x)\sin(\pi x)dx = 1/2; \quad \text{(A.32)}$$

This expression is analogous to saying the $\hat{\mathbf{i}}\cdot\hat{\mathbf{j}}=0$ and $\hat{\mathbf{i}}\cdot\hat{\mathbf{i}}=1$. If we were going to be more precise we would define the basis functions such that the inner product with itself was unity, but the factor of $1/2$ gives us no problem at this time.

So now, following our analogy to find the coefficients in front of each basis function we take the inner product of the whole expression with the first basis function ($n=1$) just like in equation A.27

$$\int_0^1 [A_1\sin(\pi x)+A_2\sin(2\pi x)+A_3\sin(3\pi x)+A_4\sin(4\pi x)+...=f(x)]\,sin(\pi x)dx \quad \text{(A.33)}$$

Since the basis functions are orthogonal, the inner product is zero for any combination of unlike basis functions. Our analogous expression to A.29

$$A_1 = 2\int_0^1 f(x)\sin(\pi x)dx. \quad \text{(A.34)}$$

For any coefficient, n, we would obtain

$$A_n = 2\int_0^1 f(x)\sin(n\pi x)dx. \quad \text{(A.35)}$$

The term on the right $\int f(x)\sin(n\pi x)dx$ is the inner product of the basis function with the known function $f(x)$. Just like $\mathbf{F} \cdot \hat{\mathbf{i}}$ gives the projection of F in the x direction, $\int_0^1 f(x)\sin(\pi x)dx$ gives the projection of $f(x)$ on $\sin(\pi x)$.

Appendix B
Thermodynamics

Thermodynamics is a course in itself. The concepts in thermodynamics are deep and far reaching. While the concepts can describe many phenomena of our physical world, thermodynamics can also be obscure, abstract and difficult to grasp. I think the best quote about thermodynamics is credited to Arnold Sommerfeld, one of the pioneers of quantum mechanics:

"Thermodynamics is a funny subject. The first time you go through it, you don't understand it at all. The second time you go through it, you think you understand it, except for one or two small points. The third time you go through it, you know you don't understand it, but by that time you are so used to it, it doesn't bother you any more."

In this appendix I will introduce you to a few key ideas. If you have already taken a course in thermodynamics, this should all seem familiar. If you have not, you should do so. In this chapter we consider only a single phase simple fluid; we are not going to worry about mixtures or phase change (i.e. from liquid to gas). I will try to keep the ideas simple but if they seem complicated you can use Sommerfeld's quote as comfort that you are not alone.

B.1 Equations of state

Three thermodynamic variables that we can directly measure are temperature, pressure, and density. These variable are intensive, meaning they are independent of the mass of the system. Extensive properties are dependent on the mass of the system; total volume V is an example.

These thermodynamic variables are not independent of each other. Imagine a piston cylinder device filled with a fixed amount of gas. You should be able to convince yourself that you could build an experiment that sets the temperature and pressure inside the cylinder, but then the volume (and thus the gases density) would have to be free to change. Likewise, you could use a fixed volume container (fixing the density) and set the temperature, but then the pressure inside would be out of your control.

The relationship that connects the thermodynamic variables is called an equation of state. This relationship, ultimately, comes from experiment where one sets two 2 of the intensive thermodynamic variables and measures the third. The simplest equation of state that we can write as a simple formula is one that you already know, the ideal gas law,

$$PV = n\overline{R}T,$$

where n is the total number of moles and $\overline{R} = 8.314\ \mathrm{J/mol \cdot K}$ is the universal gas constant. The law can also be written in intensive form as,

$$P = \rho RT,$$

where R is gas constant specific to the substance and defined as $\overline{R}/M$, where M is the molecular weight; R has units of $\mathrm{J/kg \cdot K}$. While having a simple relationship that we can write down is handy for calculations, all substances have some relationship $P = P(T, \rho)$ that can be measured. Note that in thermodynamics it is also common to use the specific volume, v, which has units of $\mathrm{m^3/kg}$. Specific volume is the inverse of the density. Thus, you may encounter the ideal gas law written as

$$Pv = RT.$$

I should add that the ideal gas law can be derived from the kinetic theory of gases, considering that the gas is made of non-interacting particles. However, the resulting equation of state is macroscopic in that there is no information about the molecular nature of matter embodied in the equation of state. Consistent with this course the equation of state is a continuum concept; however a molecular model can be used to derive or predict that equation.

B.2 Energy

There are a two important laws of thermodynamics, which you may already be familiar with. These laws are not provable, but they have stood the test of time and there are no exceptions. If you find an exception you are wrong or you will win a Nobel prize (smart money is on the former). The First Law of Thermodynamics is a statement of conservation of energy. The Second Law of Thermodynamics deals with a concept called entropy, but more intuitively the second law states that there are many processes that are not reversible in time. You cannot always put things back to the way they were. You and I will get older, you cannot unscramble an egg, work lost to friction cannot be recovered, and heat cannot flow from cold to hot. We will not deal with the second law in this course.

So the First Law has to do with conservation of energy. So what is energy? You probably know that energy comes in different forms. From particle mechanics you are likely familiar with kinetic ($\frac{1}{2}mv^2$) and potential (mgh) energy. You probably also are familiar from physics with the conservation of energy concept. If you drop something from rest and want to know how fast it is when it hits the ground, just equate the potential and kinetic energy to get $v = \sqrt{2gh}$. You also know that energy can be thermal. We discussed the idea of thermal energy in the conduction chapter. Something hot has more energy than something cold.

You probably also know that we can also convert from thermal to mechanical energy and vice verse. If I drop a rock from rest with potential energy mgh from the surface of a pond to the bottom, it seems reasonable to say that due to drag, the velocity when the rock hits the bottom will not be $v = \sqrt{2gh}$, but something much less. The lost mechanical energy has gone into motion of water in the wake of the falling rock and into viscous drag along the surface of the rock. When the water in the pond has finally come to rest, it is viscosity which has removed energy from the system and thus the lost mechanical energy has gone into thermal energy by heating up the water ever so slightly. This is the concept that we discussed in the turbulence chapter. Heating by viscosity is analogous to warming your hands by rubbing them together rapidly.

You should also be familiar with the idea of converting thermal en-

ergy to mechanical energy. This process is essentially how we derive most of our energy needs throughout the world. We burn fuel, converting chemical energy locked up in fuel into thermal energy, this thermal energy then drives the engine. This is, in at least a simplistic view, how power plants and cars work.

So I gave some examples, but you may notice I never answered the question, what is energy? When I was a student I found the idea of conservation of energy to be intuitive. I thought I understood energy. It turns out I was wrong in thinking I understood. It was not until I read the essay on energy found in Feynman's *Lectures on Physics*, that I really appreciated that most physics books never tell you what energy is. While the concept of energy seems quite real and physical, nothing could be further from the truth. Energy isn't really anything concrete but it is an abstract, mathematical concept.

The conservation of energy statement says only that there exists a number that we can compute. We happen to call that number "energy". We have rules for computing this number such as mgh or $1/2mv^2$. Conservation of energy is a mathematical statement that if something happens in the world, and we compute the energy number again it will be the same. Energy is a nothing more than a number, a mathematical function.

If we want to calculate the energy, we must be diligent. We have to be careful that we account for any energy that has come in or out of the system we are watching. Energy can change forms so we must be sure we have accounted for all the different types. If we calculate our number properly, the number will always remain the same. Conservation of energy seems intuitive, but it is actually quite abstract. You should consult the Feynman *Lectures on Physics* for a more eloquent discussion, or the book *Understanding Thermodynamics* (VanNess (1969)) which also has excellent discussions.

B.3 The First Law of Thermodynamics

The first law says that if we have a system and we put heat Q into it and extract work W from it, then the energy E of the system changes as,

$$\text{Change in } E = Q - W.$$

We are not going to worry about chemical energy, nuclear energy, mass energy or anything other than thermal energy. We will refer to E as the internal energy which has units of Joules. Technically, if we keep with the idea that energy is abstract perhaps I should refer to E as the internal energy *function*, but that language seems too lengthy (see VanNess (1969)). It is common to discuss the intensive internal energy, energy per unit mass, e in J/kg. Note that in many thermodynamics textbooks, the symbol U and u are used for the extensive and intensive internal energy. However, we reserved u for velocity in this text; so fluids people often use E and e. By default capital letters refer to extensive properties and lower case letters to intensive. Note that when we wrote the first law, we have arbitrarily decided that positive heat flows into the system and positive work comes out of the system. This sign convention is typical because of the connection to heat engines.

If we consider a closed piston cylinder device with our fluid on the inside, the work is easy to calculate. Work is defined as $W = \int F dx$ where F is the force and dx is the displacement. For a piston, the force is the pressure times the area, thus,

$$W = \int PAdx = \int PdV,$$

where dV is the change in volume of the cylinder. While we will usually consider an arbitrary material point of fluid (not a piston), the work to compress it will still be given by PdV. We can see that we have the sign right, by considering an piston cylinder insulated such that no heat escapes,

$$\text{Change in } E = -\int PdV \qquad \text{for an insulated system.}$$

If we compress the system, dV is negative and the energy goes up.

We must recognize that thermodynamics is an equilibrium subject. There is no notion of time in thermodynamics. Thus when we talked of compressing a substance in a piston cylinder device, we mean that we are compressing things so slowly that the system is always in quasi-equilibrium.

Work is a path dependent process. If we draw a Pressure-Volume diagram of some process of our piston cylinder device, the work is the area under the curve. The path that we use to connect two points changes the amount of work. The same path dependence holds for the

heat flow. When discussing differential changes, we distinguish between those that are path dependent and those that are not. The typical notation is that

$$\delta \text{ is path dependent}$$

and

$$d \text{ is path independent}$$

.

The first law written in extensive differential form is,

$$dE = \delta Q - \delta W.$$

The work W is path dependent, the heat Q is path dependent, but their difference, E, is the internal energy and is not path dependent. Anything that is not path dependent in thermodynamics is a property. Once the thermodynamic state is set (say by setting pressure and temperature) then the internal energy is set as well. It does not matter the path we took, the energy only depends on the local thermodynamic state. The First Law is really interesting in that is says the difference of these two path dependent things results in something which is not path dependent.

In intensive form, we can write the first law as

$$de = \delta q - \delta w,$$

or substituting for the PdV work,

$$de = \delta q - Pdv.$$

This equation gives us the framework to measure the *mathematical* function e as a function of the measurable thermodynamic variables. Since we believe the First Law so much, we can use the law itself to calculate the internal energy function. Once we know the internal energy function for a substance, we can use the First Law to calculate the work or the heat for some other process. If this circular logic bothers you (which came first, the First Law or the internal energy function), I suggest you just go with it until it doesn't bother you anymore.

B.4 Specific heats

From calculus, we know that if something is a function of many variables, $x(a, b, c)$, the total derivative is written as

$$dx = \left(\frac{\partial x}{\partial a}\right)_{b,c} da + \left(\frac{\partial x}{\partial b}\right)_{a,c} db + \left(\frac{\partial x}{\partial c}\right)_{a,b} dc$$

In the notation we take the partial derivative of x with respect to each variable, holding the others constant. The subscript outside the parenthesis denote that those variables are held constant.

Since in thermodynamics of a homogeneous substance, any property is a function of any two other properties, we could say for example that internal energy is a function of temperature and volume, $e(T, v)$, thus the change in internal energy is given by,

$$de = \left(\frac{\partial e}{\partial T}\right)_v dT + \left(\frac{\partial e}{\partial v}\right)_T dv. \tag{B.1}$$

Let's consider a process where we add a quantity of heat (per unit mass) of δq while holding the volume constant. This First Law tells us that

$$de = \delta q - P dv,$$

however since the volume is constant then no work is done and,

$$de = \delta q.$$

From Equation B.1 for a constant volume process ($dv = 0$), calculus tells us that

$$de = \left(\frac{\partial e}{\partial T}\right)_v dT.$$

Equating the last two expression gives

$$\frac{\delta q}{dT} = \left(\frac{\partial e}{\partial T}\right)_v \equiv c_v.$$

The ratio $\delta q/dT$ is called the constant volume specific heat. It is the amount of heat added to a substance to change its temperature a degree while holding the volume constant. We have discussed the specific heat before, but now we are being more precise. The specific heat must be measured holding something constant, in this case the volume. The

specific heat is something we can measure and it is equivalent to the derivative of the internal energy function with respect to temperature.

We can also measure the specific heat at constant pressure. The analysis here is only a little bit trickier. Since the first law tells us that $de = \delta q - Pdv$, when the pressure is constant we can rewrite as

$$d(e + Pv) = \delta q.$$

The quantity $e + Pv$ shows up so often in thermodynamics, it gets its own name enthalpy, h. It is perfectly legitimate to combine properties in any way we please to get a new property. While this seems a mathematical trick, remember that energy is just a mathematical function, thus it is no more abstract to define a new function, the enthalpy. Defining the enthalpy, we can write for a constant pressure process,

$$dh = \delta q.$$

Since h is a property we can write $h(T, P)$ and use calculus to get,

$$dh = \left(\frac{\partial h}{\partial T}\right)_P dT + \left(\frac{\partial h}{\partial P}\right)_T dP.$$

The last term drops for a constant pressure process. Equating the last two equation we get an expression to calculate the constant pressure specific heat.

$$\frac{\delta q}{dT} = \left(\frac{\partial h}{\partial T}\right)_P \equiv c_p.$$

The difference between the constant volume and constant pressure specific heat is that in the constant pressure case some work is done in the expansion/contraction of the material.

In the case of liquids and solids, c_p and c_v are essentially equal. For gases they are not.

B.4.1 Ideal Gas

For an ideal gas, it turns out that the internal energy is a function of temperature only; i.e. $e(T)$. In this case, the specific heats,

$$c_v = \frac{de}{dT}; \qquad c_p = \frac{dh}{dT},$$

are given as simple differentials. For an ideal gas it is pretty easy to prove that

$$c_p - c_v = R$$

where R is the gas constant specific for the material.

Calculating the change in energy between two temperatures is then given by,

$$e_2 - e_1 = \int_{T_1}^{T_2} c_v dT. \tag{B.2}$$

In practice, the specific heat is approximately constant if the temperature changes are not too great and we have a very simple integral,

$$e_2 - e_1 = c_v(T_2 - T_1). \tag{B.3}$$

Appendix C
Energy equation

C.1 Conservation law

The first law of thermodynamics states that the rate of change of the total energy of a system must equal the net rate of heat flowing into/out of the material and the work done on it. This law must hold true for every fluid particle in our system. If we consider an arbitrary fluid particle we must consider heat flowing in and out of that particle, forces acting on the surface of the particle, and mechanical work done by the body forces. In integral form, the first law of thermodynamics is,

$$\frac{d}{dt}\int_{V(t)} \rho(e+\frac{1}{2}\mathbf{v}^2)dV = -\int_{S(t)} \mathbf{q}\cdot\mathbf{n}dS + \int_{S(t)} \mathbf{s}\cdot\mathbf{v}dS + \int_{V(t)} \rho\mathbf{v}\cdot\mathbf{f}_b dV$$

The term on the left side is the rate of change of the energy (internal plus kinetic). The terms on the right hand side refer to the net rate of heat flowing in/out of the volume, the work done by surface forces, and the work done by body forces. The vector $\mathbf{s}$ refers to the stress vector at a point on the surface of the arbitrary material volume and the product $\mathbf{s}\cdot\mathbf{v}$ is mechanical power at the surface of the fluid particle; force times velocity. Recall the that the stress vector is related to the stress tensor by $\mathbf{s} = \mathbf{n}\cdot\mathbf{T}$. The product $\mathbf{v}\cdot\mathbf{f}_b$ is the body force vector dotted with the velocity.

What is new in this formulation from what we considered in the conduction chapter is 1) that the material volume is changing with time, 2) the energy is composed of kinetic and internal energy, and 3) mechanical work can be done. Using the Reynolds Transport Theorem

on the left hand side, we would obtain,

$$\int_{V(t)} \rho \frac{D(e+\frac{1}{2}\mathbf{v}^2)}{Dt} dV = -\int_{S(t)} \mathbf{q}\cdot\mathbf{n} dS + \int_{S(t)} \mathbf{n}\cdot\mathbf{T}\cdot\mathbf{v} dS + \int_{V(t)} \rho\mathbf{v}\cdot\mathbf{f}_b dV$$

We can convert the surface integrals to volume integrals on the right hand side using the divergence theorem to obtain,

$$\int_{V(t)} \left(\rho \frac{D(e+\frac{1}{2}\mathbf{v}^2)}{Dt} = -\nabla\cdot\mathbf{q} + \nabla\cdot(\mathbf{T}\cdot\mathbf{v}) + \rho\mathbf{v}\cdot\mathbf{f}_b \right) dV.$$

As before, since the integral equation is true for any arbitrary material volume, the integrand is zero at all points in space. Thus we arrive at conservation of energy in differential form,

$$\rho \frac{D(e+\frac{1}{2}\mathbf{v}^2)}{Dt} = -\nabla\cdot\mathbf{q} + \nabla\cdot(\mathbf{T}\cdot\mathbf{v}) + \rho\mathbf{v}\cdot\mathbf{f}_b.$$

Recall that the total stress tensor can be written, for a Newtonian, incompressible flow, as $\mathbf{T} = P\mathbf{I} + 2\mu\mathbf{S}$, where $\mathbf{S}$ is the strain rate tensor. Thus, for a Newtonian fluid where gravity is the only body force,

$$\rho \frac{D(e+\frac{1}{2}\mathbf{v}^2)}{Dt} = -\nabla\cdot\mathbf{q} - \nabla\cdot(P\mathbf{v}) + \rho\mathbf{v}\cdot\mathbf{g} + 2\mu\nabla\cdot(\mathbf{S}\cdot\mathbf{v}). \quad \text{(C.1)}$$

This equation, looks pretty complicated to me and we will never use conservation of energy in this form. We can convert it to a much simpler and useful form which is easier to interpret.

C.2 Thermal energy equation

The first simplification of the energy equation is to split the equation into a mechanical and thermal part. Before we proceed it is useful to remind a simple result from particle mechanics, just to remind you that some of the manipulations we make in this chapter are not that unusual or arbitrary. Consider a particle of constant mass m in a gravitational field with no other forces. The acceleration due to gravity will be a constant g pointing downward in the z direction. Newton's Laws would say that

$$m\frac{dv}{dt} = -mg.$$

If we multiply the above equation by the velocity v we obtain,

$$v\frac{d\ mv}{dt} = -mgv.$$

We can re-write the left side and noting that velocity is the time rate of change of position, $dz/dt = v$,

$$\frac{d\ \frac{1}{2}mv^2}{dt} = -mg\frac{d\ z}{dt}.$$

which we can rearrange to obtain,

$$\frac{d}{dt}\left(\frac{1}{2}mv^2 + mgz\right) = 0.$$

This statement should be familiar to you and says that the kinetic plus potential energy of the system is a constant. We often describe kinetic and potential energy to be forms of mechanical energy.

We can follow the same procedure to find the mechanical energy contained in a fluid flow. Namely, we take the dot product of the momentum equation with the velocity vector,

$$\left(\rho\frac{D\mathbf{v}}{Dt} = -\nabla P + \rho\mathbf{g} + 2\mu\nabla\cdot\mathbf{S}\right)\cdot\mathbf{v}.$$

which can be expanded out to obtain,

$$\rho\frac{D\frac{1}{2}\mathbf{v}^2}{Dt} = -\nabla P\cdot\mathbf{v} + \rho\mathbf{g}\cdot\mathbf{v} + 2\mu(\nabla\cdot\mathbf{S})\cdot\mathbf{v}.$$

The mechanical energy can be subtracted from the total energy equation, Equation C.1. The result is an equation for only the thermal energy. It's fine to subtract equations from each other since we are removing a piece of the equation that is already satisfied (i.e. momentum). Skipping quite a few details of expanding out the vector operators, the resulting thermal energy equation is,

$$\rho\frac{De}{Dt} = -\nabla\cdot\mathbf{q} - P\nabla\cdot\mathbf{v} + 2\mu\mathbf{S}:\nabla\mathbf{v}. \tag{C.2}$$

The operator ":" is a new one for you. It is like a tensor dot product. You simply multiply each entry of the tensor by the respective entry of the other tensor and sum them all up. The equation reads as follows. The left side is the material derivative of the internal energy of a fluid particle as it moves about. This internal energy goes up or down based

on the right hand side. The first term on the right side is the divergence of the heat flux. This term is just like we found in the conduction problem. The second term is energy going up or down due to compression. Recall that $\nabla \cdot \mathbf{v}$ is a measure of the fluid being locally compressed. If you compress something, it heats up. The last term is heat generated by viscous dissipation, or friction. It turns out that you can prove (via the Second Law of Thermodynamics) the last term always increases the thermal energy. If you shook up a bottle of fluid and let it come to rest by viscosity, the kinetic energy of the fluid in the bottle has to go somewhere, it goes into heating up the fluid.

In practice, it is more convenient to work with temperature as a variable rather than internal energy. We can get the equation in terms of temperature by 1) applying Fourier's law for the heat flux $q = -k\nabla T$, and 2) using thermodynamics to relate internal energy to temperature. This is the point where the subtleties really come in. However, for an incompressible liquid, everything is simple. The divergence of the velocity field is zero, $\nabla \cdot \mathbf{v}$ and a constant heat capacity, C, is a good thermodynamic model of most liquids; i.e. $de = CdT$. For a liquid we don't need to worry about the constant volume and constant pressure specific heats. Therefore for liquids the energy equation is,

$$\rho C \frac{DT}{Dt} = \nabla \cdot (k\nabla T) + 2\mu \mathbf{S} : \nabla \mathbf{v}; \quad \text{[for an incompressible liquid]}.$$

This equation is what we guessed on the first page of the chapter except we found there is a viscous dissipation term which converts kinetic energy lost due to viscosity into heat.

C.2.1 Thermal energy equation, gases

As a disclaimer, I should state that the details of this section are not all that important for our purposes in this course. We would need more time and more detailed knowledge of thermodynamics for this section to really make sense. However, I do want to include a little bit of detail, just so you are aware that there are some subtleties.

Now let's return to the thermal energy equation,

$$\rho \frac{De}{Dt} = \nabla \cdot k\nabla T - P\nabla \cdot \mathbf{v} + 2\mu \mathbf{S} : \nabla \mathbf{v},$$

and consider the case of gases. Now you might be tempted to still throw

out the term with $\nabla \cdot \mathbf{v}$ for an incompressible gas flow. However there is a bit of a subtlety here if you are dealing with gas flows that are nominally incompressible. The problem is that while the velocity field has nearly zero divergence, $P\nabla \cdot \mathbf{v}$ is not necessarily small compared to the other terms in the equation.

From the thermodynamics chapter, we know that the internal energy can be written as a function of the other variables, let's use $e(T, \rho)$. Thus,

$$de = \left(\frac{\partial e}{\partial T}\right)_{\rho} dT + \left(\frac{\partial e}{\partial \rho}\right)_{T} d\rho = c_v dT + \left(\frac{\partial e}{\partial \rho}\right)_{T} d\rho.$$

So in our energy equation we replace the differential change with the material derivative, i.e.

$$\frac{De}{Dt} = c_v \frac{DT}{Dt} + \left(\frac{\partial e}{\partial \rho}\right)_{T} \frac{D\rho}{Dt}.$$

Using our general conservation of mass we can rewrite as

$$\frac{De}{Dt} = c_v \frac{DT}{Dt} - \rho \left(\frac{\partial e}{\partial \rho}\right)_{T} \nabla \cdot \mathbf{v}.$$

Substituting into our original energy equation yields,

$$\rho c_v \frac{DT}{Dt} = \nabla \cdot k\nabla T - \left(P - \rho \left(\frac{\partial e}{\partial \rho}\right)_{T}\right) \nabla \cdot \mathbf{v} + 2\mu \mathbf{S} : \nabla \mathbf{v},$$

There are a variety of thermodynamic relationships that are derived based on multi-variable calculus and the First Law. Using one of these relationships (see Bird et al. (1960) for details) we can rewrite the equation as,

$$\rho c_v \frac{DT}{Dt} = \nabla \cdot (k\nabla T) - T \left.\frac{\partial P}{\partial T}\right|_{V} \nabla \cdot \mathbf{v} + 2\mu \mathbf{S} : \nabla \mathbf{v}. \tag{C.3}$$

Which is a general statement of conservation of energy for a Newtonian fluid in terms of temperature. The quantity, $\left.\frac{\partial P}{\partial T}\right|_{V}$ comes from the equation of state This form is useful as all the variables are measurable thermodynamic properties. Don't worry that the steps which make use of thermodynamic property manipulations seem mysterious; it is just a trick that was worked out.

Using some more relationships from thermodynamics and conservation of mass, we can transform the equation to another equivalent,

$$\rho c_p \frac{DT}{Dt} = \nabla \cdot (k \nabla T) + T\beta \frac{DP}{Dt} + 2\mu \mathbf{S} : \nabla \mathbf{v}. \tag{C.4}$$

Here c_p is the constant pressure specific heat and $\beta = -\frac{1}{\rho} \left. \frac{\partial \rho}{\partial T} \right|_P$ is called the bulk expansion coefficient. The bulk expansion coefficient comes from the equation of state and is a property that can be looked up. Equations C.2, C.3, and C.4 are all formally equivalent statements of conservation of energy. Even though the details of each step might seem mysterious, we are doing nothing other than using thermodynamics to change variables to a more useful and equivalent form of the equation. Again, don't get caught up in the details of the manipulation.

For an ideal gas, $T\beta = 1$, thus,

$$\rho c_p \frac{DT}{Dt} = \nabla \cdot (k \nabla T) + \frac{DP}{Dt} + 2\mu \mathbf{S} : \nabla \mathbf{v}; \quad \text{[for an ideal gas]}.$$

In many cases of gas flows the term DP/Dt is relatively small and can be discarded. The size of the term can be estimated by comparing the left hand side to the term DP/Dt. In a flow the pressure scale, dP, is related to the velocity scale through $dP \sim \rho U_0^2$. Thus if $U_0^2/(c_p \Delta T)$ is small, then you are fairly safe in discarding the pressure term. This is often the case as c_p is usually on the order of 1000 J/kg K for gases. Under the assumption of a nearly constant pressure flow with an ideal gas, the simplified energy equation is,

$$\rho c_p \frac{DT}{Dt} = \nabla \cdot (k \nabla T) + 2\mu \mathbf{S} : \nabla \mathbf{v}; \quad \text{[for a gas at constant P]},$$

The equation above is actually different than Equation C.3 if we simply discarded the $\nabla \cdot \mathbf{v}$ term from the start. In a gas the constant pressure and constant volume specific heats are not the same and in an ideal gas $c_p - c_v = R$.

Bibliography

Anderson, J.D. 2005. Lundwig Prandtl's boundary layer. Physics Today, **58**, 42–48.

Bird, R.B., Stewart, W.E., and Lightfoot, E.N. 1960. Transport Phenomena. Wiley.

Bridgman, P.W. 1922. Dimensional Analysis. Yale University Press.

Incroprera, F.P., and DeWitt, D.P. 2001. Fundamentals of Heat and Mass Transfer. Wiley.

Kundu, P.K., and Cohen, I.M. 2004. Fluid Mechanics. Elsevier Academic Press.

Panton, R.L. 1996. Incompressible Flow. Wiley.

Rayleigh. 1915. The principle of similitude. Nature, **95**, 66–68.

Reynolds, O. 1883. An experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous and the law of resistance in parallel channels. Phil. Trans. R. Soc., **174**, 935–982.

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

Schey, H.M. 1997. div grad curl and all that. W.W. Norton and Co.

VanDyke, M.V. 1982. An Album of Fluid Motion. Parabolic Press.

VanNess, H.C. 1969. Understanding Thermodynamics. Dover.