


The background of the entire page is a painting of a sunset over the ocean. The sun is a bright, glowing orb in the center of the horizon, surrounded by thick, horizontal bands of orange, red, and yellow clouds. The sky above the clouds is a mix of soft, blended colors like green, blue, and orange. The ocean below the horizon is dark and textured with blue and brown tones. In the lower-left quadrant of the image, a small, dark silhouette of a person is visible in the water.

# COMMENTARY ON FLUID MECHANICS

*Arnaldo Rodriguez-Gonzalez*

Copyright © 2020 by Arnaldo Rodriguez-Gonzalez. Second edition. This work is licensed under a Creative Commons Attribution 4.0 International License. 

Cover art: *La Víspera Del Hombre (The Eve of Man)*, Arnaldo Rodriguez-Gonzalez.

First edition written in 2019. The author invites readers to store and distribute this text freely, and to appropriately cite this text when making reference to it. Usage of artwork not created by the author in this text is purely for analytical purposes in service of pedagogy, and is in line with the CAA's *Code of Best Practices in Fair Use for the Visual Arts*—as such, its usage is intended to be covered by fair use. Use of this text for commercial purposes or resale is strongly discouraged by the author and could violate fair use as described above. A modified online version of this text is available [here](#). Text may be updated periodically for error corrections.

# PREFACE

On April 17, 1969, in the height of the Cold War, famed American accelerator scientist Robert Wilson testified in front of the Joint Committee on Atomic Energy to seek funding for the National Accelerator Laboratory's mission to study subatomic particles; the building blocks of nature. When asked by the committee about the value of the knowledge we'd obtain from this program with respect to the arms race rapidly forming between the Soviets and the Americans, Wilson responded: "This new knowledge has nothing to do with defending our country except to help make it worth defending."

Many nights while writing this book, I asked myself: "Why am I writing this? What could I, a measly Ph.D. candidate, possibly contribute to the pedagogy of fluid mechanics? Why am I not focusing on things that will make me obtain my degree faster?" The answer is not that my writing contains some mysterious kernel of unknown

knowledge, or that this monograph is some kind of pedagogical miracle. It is, as Wilson noted, that some things are done for the sake of making other things worth doing, and that certainly holds true for this document. I didn't choose to obtain a Ph.D. to gain some abstract social credential or a cushy job, but to learn the inner workings of Nature; and to share that knowledge freely and openly with the curious & interested. In that sense, writing this before I obtain my doctorate means I am simply getting ahead of myself a little bit—I hope you, the reader, won't fault me for it.

## **Scope & Structure**

This book is the result of a series of lecture notes I wrote while serving as a teaching assistant for the introductory fluid mechanics course at Cornell, designed to be read as a complement for introductory learners of fluid mechanics alongside a more generalized text—many of which you may find in the bibliography section at the end of the text. It was created, in part, to address the questions I saw most often from my students that the canon of introductory fluid mechanics textbooks couldn't answer. What is viscosity, really? Why are the Navier-Stokes equations so difficult to solve, and how do you derive them? Why is drag sometimes linear and sometimes quadratic, but never cubic? In any case, I hope you will find my answers to these

questions satisfactory.

I begin by discussing the conceptual nature of what we call a fluid, and how it emerges from collections of uncountably large numbers of molecules; followed by a description of the key properties associated with such fluids which naturally emerge from such a description. I then discuss the mechanics of unmoving fluids, the role pressure plays in those mechanics, and how pressure & gravity generate buoyancy forces on objects within fluids. Afterwards, I discuss the mathematical tool of control volume analysis, which allows for a rigorous description of the effect of force fields on the surfaces of immersed objects, and motivate its importance by using it to derive general laws of motion for fluids within closed channels, a subfield of fluid mechanics known as hydraulics.

Afterwards, I discuss a generalization of control volume analysis called transport theory, which allows us to formulate fundamental conservation laws of mass and momentum for fluids. After a brief intermezzo where I discuss the notion of a tensor non-rigorously, I connect the conservation of momentum law within a fluid to the celebrated Navier-Stokes equations by exploring the connection between the properties of the microscopic particles making up a fluid and the properties of the fluid themselves—a study also known as rheology. With the Navier-Stokes equation in tow, I discuss its pathology and why it is usually impossible to solve analytically, and demonstrate



this by deriving the equations of motion for flow within pipes and show how they break down as a result of turbulence. I then introduce the idea of dimensional analysis with the motivation of developing estimates of the onset of turbulence, and show that the Reynolds number governs the onset of turbulence in pipe flow.

To conclude, I discuss steady flow past immersed objects and flow past steadily moving objects, and form approximations to solutions of the Navier-Stokes equations both far from the object (the far field) and near to it (the near field). While doing so, I derive the general mathematical forms for irrotational/potential flow as well as for Stokes/creeping flow, and show the emergence of "paradoxes" resulting from these approximations. Finally, I discuss the flow in the region in between the near & far field, referred to sometimes as the boundary layer, categorize the stages of turbulence in this type of flow, and derive universal laws of lift & drag for objects of an arbitrary shape using dimensional analysis.

Each chapter is prefaced by artwork related to the fluid-mechanical concepts contained in that chapter, presented to invite the reader to analyze it in the context of the chapter's content. Each chapter also contains conceptual motivating questions, as well as commentary comparing the subject coverage in this text to other texts in the literature and pointing the reader to more specific, detailed texts on the chapter's material.

# CONTENTS

o1 Fluid Concepts	1
o2 Pressure & Hydrostatics	10
o3 Buoyancy	19
o4 Control Volume Analysis	29
o5 Hydraulics	42
o6 Transport Theory	55
o7 Intermezzo: Tensors	66
o8 Rheology	76
o9 Navier-Stokes, Existence & Uniqueness	91
10 Pipe Flow & Turbulence	104

11	Dimensional Analysis	119
12	Far Field & Irrotational Flow	139
13	Near Field & Creeping Flow	160
14	Boundary Layers, Lift, & Drag	174
	Bibliography	192



# O1

## FLUID CONCEPTS



*"The Beach at Heist", Georges Lemmen.*

Before delving into the study of fluid mechanics—or the study of anything, really—it's imperative to look at the underlying assumptions we make when we try to analyze the phenomena we wish to learn about. Namely, what is a fluid, in a conceptual/philosophical sense? And how does that description of a fluid create other related concepts that we can observe and measure?

For starters, most of the matter we interact with on a daily basis tends to come in incomprehensibly large clumps of smaller building blocks called atoms or molecules. These clumps can be either mostly stiff and with a specific shape, in which case we would usually describe the clump as solid, or flowing and with uncertain, changing forms, which we would describe as fluid.

In either case, trying to understand the behavior of a single clump really entails understanding the collective behavior of a hundred sextillion of these building blocks, each performing their own complicated dance through time and space. And if you think that trying to understand what each of these molecules is doing is effectively impossible, I'd agree with you! Even though it would be possible to describe the physics of each of these constituents fairly straightforwardly using models of molecular physics, not even the most powerful supercomputer would be able to easily and faithfully obtain the motion of these molecules from them due to their sheer numbers.

However, this incomprehensible complexity runs contrary to

most of our daily experiences. I don't expect my morning coffee to spontaneously crawl up the side of my cup and spill itself—and if I were to tilt my cup of coffee, I would reliably see its liquid surface stay parallel to the ground. Water from the faucet usually comes out in a steady stream, ketchup bottles doesn't spontaneously explode or dissolve, and the ocean mostly stays put where it's usually been. In short, the minuscule collective randomness we would expect to see from these massive assemblies of molecules averages out into behavior that is fairly uniform and easy to understand at the scales that we can see and feel.

As a result, the theory of fluids (and solids) is a theory that only cares about those scales that we can directly experience, and considers those tiny building blocks only when it really needs to. This means that the way we mathematically describe large clumps of fluid matter is as precisely that—continuous clumps. Formally, we refer to the theory of fluid mechanics as a continuum theory.

In a continuum theory, we usually focus on some abstract blob of “something” (in this case, matter) distributed over space, which has some properties that also vary over the space the blob occupies. For example, the atmosphere can be thought of as a moving continuum “blob” of air (and other chemicals) whose speed and density changes as you move around and above the Earth. But this brings up another question; what properties do we generally care about in a fluid?

Since the distinguishing characteristic of fluids is motion, surely you'll agree that velocity is a property we care about. In the theory of fluid mechanics, we associate each point in our 3-D space  $(x, y, z)$  in a fluid at a time  $t$  with a velocity  $\vec{v}(x, y, z, t)$ . This fluid velocity has three distinct components,  $v_x(x, y, z, t)$ ,  $v_y(x, y, z, t)$ , and  $v_z(x, y, z, t)$ , each representing the speed of the fluid point in the  $x$  /  $y$  /  $z$  direction. Its interpretation is straightforward; we expect a "point" of fluid located at  $(x, y, z)$  to move with velocity  $\vec{v}(x, y, z, t)$  at time  $t$ . But what does a "point" of fluid even mean?

Recall that, at the scales we experience, what we think is a "point" of fluid is actually a huge mess of molecules randomly bouncing around at a molecular scale. Therefore, in order to make any sense of this at the continuum scale, we need to somehow "smooth" out all the molecular unpredictability into something predictable and measurable at the continuum scale. Luckily, we can do this without losing too much accuracy by selecting some specific volume size that is very small at the continuum scale, and treating that volume like a continuum "point". We can then define the properties of that fluid "point" as an average of the properties of the molecules inside it. This is commonly referred to as the continuum approximation, and the properties we obtain from these averages are called continuum or fluid properties.

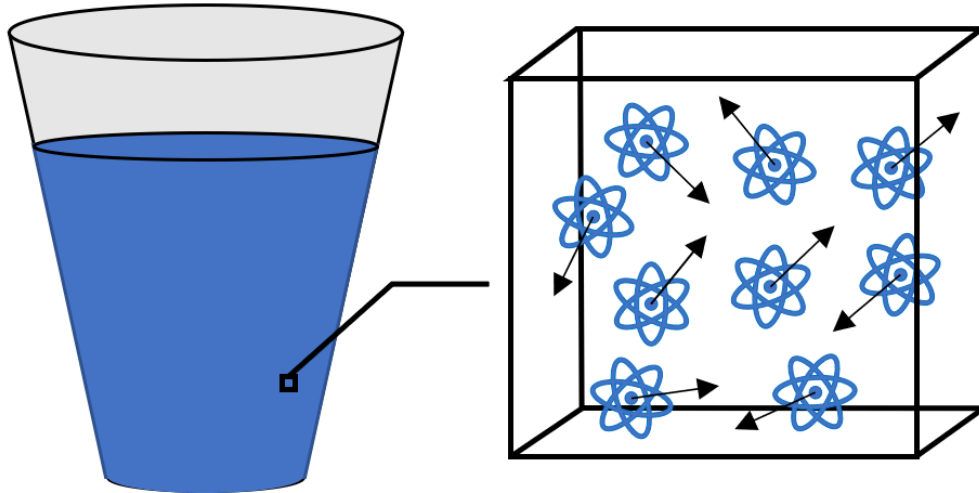


Figure 01.1: What we describe as a "point" of fluid is actually a group of many molecules moving around randomly inside a molecular-scale volume. Instead of perceiving each of these molecules's properties, we perceive a type of average of these at a given fluid "point".

To be more precise, the properties of fluids we obtain using the continuum approximation are limits of ratios; ratios of volumes and the sum of the properties of molecules inside them. To demonstrate, consider a spherical molecular "net" in a fluid through which molecules pass in and out. At every instant, the net contains some well-defined amount of fluid molecules, and therefore some well-defined total molecular mass. If this net is small at the molecular scale, random perturbations to the total fluid mass within the net as a result of molecules moving in and out of it screw up our ability to define a consistent value of total fluid mass in the net over time. However, as you make the net bigger and bigger at the molecular scale—while keeping it point-like in the continuum scale—the ran-

domness smooths itself out due to the sheer number of molecules, and one observes a total fluid mass within the net that is only a function of the net volume. The ratio of that total fluid mass to the net volume is called the density  $\rho(x, y, z, t)$  of the fluid.

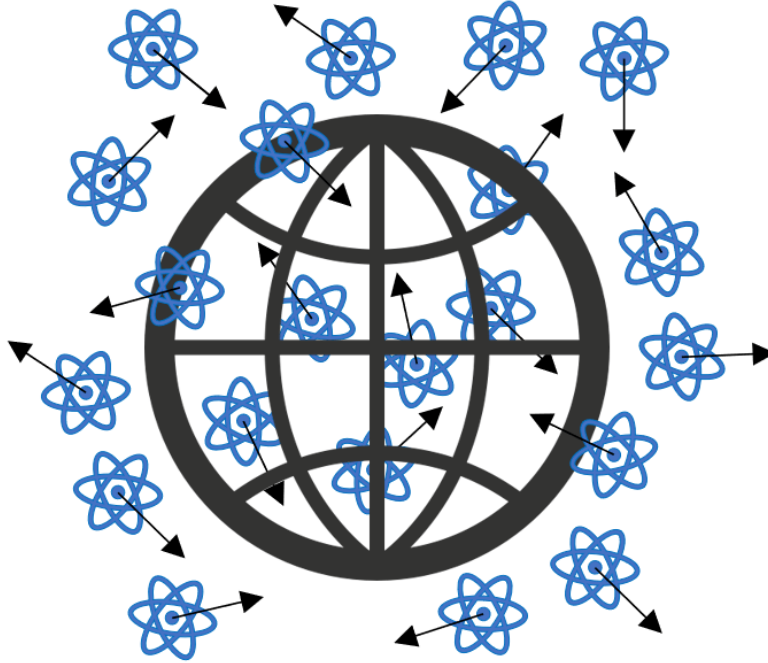


Figure 01.2: Molecules in a fluid randomly move in and out of this conceptual molecular "net", which is point-like at the continuum scale. When the net is small at the molecular scale, taking the ratio of the total molecular mass within the net and the volume of the net leads to a measurably inconsistent result due to random molecular motion. When the net is sufficiently big, this ratio settles into a consistent value we call the fluid's density.

Likewise, we could do the same thought experiment but now counting up the energy of the fluid molecules in the net, specifically considering the energy coming from the unpredictable back-and-

forth motions of molecules at the nanoscale. As the net becomes bigger, the ratio of the total energy divided by the net volume approximates a kind of molecular energy density usually referred to as pressure  $p(x, y, z, t)$ . This is unrelated to the energy from motion of fluid "points" at the continuum scale; pressure is entirely molecular.

Finally, as you might expect, the velocity of a fluid point is simply the average velocity of the molecules within a molecular net of the appropriate size. Interestingly, although each individual molecule possesses a decent instantaneous velocity from random molecular motion, it turns out that the average velocity (i.e. the fluid velocity) of a group of molecules is very often quite low in comparison. As a result, almost all molecular motion is irrelevant to fluid motion. This doesn't mean that what happens at the molecular scale is completely disconnected from what happens at the continuum scale, though; it just means that we should be able to describe most of what we see in fluid mechanics using only the averaged, continuum properties we described above.

This trinity of properties associated with points of fluid—fluid velocity, density, and pressure—are usually all we need in elementary fluid mechanics to be able to describe the motion of fluids. As a result, we expect the chief enterprise of any fluid mechanician to be describing how each of these properties is related to each other, and how things external to a fluid (forces, etc.) change them.



## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. How can you preemptively check if the continuum approximation is appropriate? Can you think of some "measure" of how good it is?
3. Can you think of other useful fluid properties you could obtain using the porous sphere technique detailed above?
4. Why does it make sense to assume that the properties of many similar, randomly-moving molecules average out to something non-random? Does this arise from a physical phenomenon, or a mathematical one?
5. How would you try to estimate the molecular energy of a group of molecules? Are there other concepts in physics & engineering that could help you do that?
6. Can you make any guesses as to how some continuum properties affect each other based on their molecular equivalents?

7. From an experimental perspective, can you think of any reason why this definition of fluid properties is convenient? How would you as an experimentalist measure them?

## Further Reading & Context

A simple physical introduction to the continuum approximation and to the distinctions between different states of matter can be found in [5], and in a more rigorous but limited sense in [34]. More advanced treatments of the connection between the molecular and macroscopic scales in fluids can be found in [46][18][15]. A comprehensive description of the use of molecular-scale physics to simulate fluids is given in [1], and a more theoretical perspective on molecular approaches to modeling fluids can be found in [25]. Nearly all fluid mechanical textbooks (see [34] [18] [47] [33]) define pressure as a force per unit area applied by a fluid rather than as an energy density, likely in order to avoid any conflicts generated by differences in definitions of pressure in advanced thermodynamical contexts—I take no such caution. See [3] or [34] for a nice introduction to such issues.

Often, scientists are interested in fluid physics occurring at a regime where macroscale fluid properties are good descriptors of behavior but where microscopic, random phenomena are apparent—such regimes are discussed well in [59][23].

# 02

## PRESSURE & HYDROSTATICS



*"Still Life with Lemon", Henri Matisse.*

If our objective as fluid mechanics is to understand how pressure, density, and velocity are related, we should then begin to look at simple examples of fluid mechanical phenomena to get a sense of how these properties interact with each other and with external forces—and in particular, how pressure and external forces interact.

Phenomenologically, it's important to note that the amount of molecular energy present in some sample of fluid molecules is humongous here on Earth—even in fluids which are completely at rest at the continuum scale. As a result, fluid molecules tend to be pretty good at distributing themselves locally in a way that minimizes that molecular energy. And thanks to all the pushing and pulling between molecules, they tend to rapidly space themselves out so consistently that we can usually always define the number of particles in a given fluid point—and quantities connected to it, like density—as an exclusive function of the molecular energy density/pressure of that point. In short,  $\rho = \rho(p)$ .

This may lead you to believe that applying a force to a fluid will lead to a change in its density. However, the amount of energy you are giving to a fluid molecule by applying a continuum-scale force to it is positively meager in comparison to the molecular energy the molecule already possesses through pressure. As a result, external forces and other continuum-scale effects almost never alter the density of a fluid—they usually just trigger continuum-scale fluid motion

precisely so that the fluid can preserve its density. This assumption or property of most fluids is usually referred to as incompressibility, and it causes the pressure to be uniquely defined solely by the requirement that the density be the same everywhere.

An everyday example of this can be found in a glass of water. Every point of water in the cup is feeling an identical downwards force density  $\vec{f}$ , proportional to the density of the point and the gravitational constant ( $\vec{f} = \rho\vec{g}$ ), which we assume is the same everywhere. For future reference, a force distributed through a fluid like this is commonly referred to as a body force. So if Newtonian mechanics holds, then how are those points of water not moving? Why doesn't all the water simply accumulate into a highly dense thin film at the bottom of the cup?

Common sense indicates that another force, namely a molecular force, needs to be countering gravity in order for the water in the cup to retain its density:  $\rho\vec{g} + \vec{f}_{molecular} = 0$ . This molecular force is coming from pressure—the molecular energy density of the water is redistributing itself to ensure the density stays the same. This redistribution causes gradients in pressure that manifest as continuum force densities, leading us to obtain the force balance equation for fluids not in motion, or the hydrostatic equation:

$$\vec{f} + \nabla p = 0 \text{ (in general)}$$

$$\rho \vec{g} + \nabla p = 0 \text{ (when the external force is gravity)}$$

Heuristically, this means that pressure increases in the direction of forces for static fluids. Noting gravity points "downwards", this is why deep-sea divers can't go too far down into the ocean without a submersible (they'd get crushed by the increased pressure/molecular energy of the water) and why astronauts wear full-body suits (our bodies' molecular energy would get dumped out into the very low-pressure environment of space).

But body forces aren't the only way forces can influence a fluid; we should also consider the influence from external forces that, rather than being distributed through a fluid like gravity, are concentrated on solid surfaces in contact with the fluid (which is how a cup holds water). These surface forces are a little trickier to interpret, but easy to describe with the right conceptual machinery.

Consider a point of fluid located right at the bottom of a cup, in contact with a tiny patch of cup. The point itself isn't moving, but the molecules within the point are—and something needs to compensate for the lack of fluid points below the one of interest to ensure the fluid point doesn't move. That compensation is coming from an increase of molecular energy inside that patch of cup to "match" the point's surroundings. That increase, and its effect in ensuring the fluid point remains stationary, can be mathematically represented as a local force per area pointing into the fluid with the

same magnitude as the local pressure ( $|\vec{t}_s| = p$ ).

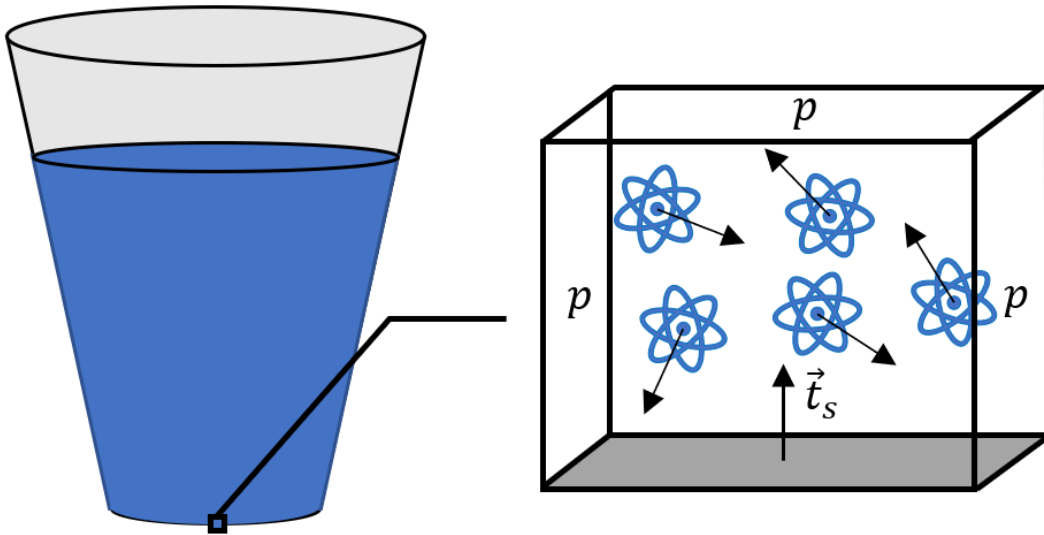


Figure 02.1: In a fluid point at the bottom of the cup, molecules under the influence of gravity bounce around against their neighboring fluid points and a tiny patch of cup. The neighboring pressure keeps the particles from wandering up or sideways, while a surface force from the cup patch keeps the particles from moving downwards due to gravity.

We can call that local force per area a surface traction, contact pressure, or simply a traction; there isn't really a standard clear name for the concept, and many sources incorrectly refer to it as pressure without making the distinction between the local force per unit area and its molecular source.

To humanity's benefit, this principle of induced surface forces due to pressure works both ways: if we impose a surface force on a fluid, a static fluid will locally increase its pressure by the magnitude



of that force per unit area to compensate. This bidirectional principle lets us manipulate forces acting on objects in useful ways, all of which are variations of the following sequence of phenomena:

1. An object imposes a net force on a fluid, in the form of a force per unit area distributed over a surface.
2. The surface force generates a pressure increase throughout the fluid.
3. The extra pressure is transmitted as a force per unit area onto another object with a different surface area, leading to a different net force acting on that second object.

The field of engineering that utilizes this force-multiplying principle to solve problems (among many other fluid-mechanical tools) is called hydraulics, and has many applications throughout varying fields of science & technology. Notable examples include hydraulic jacks, hydraulic suspensions, hydraulic presses, etcetra.

The arguments we've made here are quite general, but don't really account for what happens when the traction acting on a surface isn't the same everywhere. To understand what happens in that situation, we'll need to look at one of the first great scientific discoveries of fluid mechanics; buoyancy.

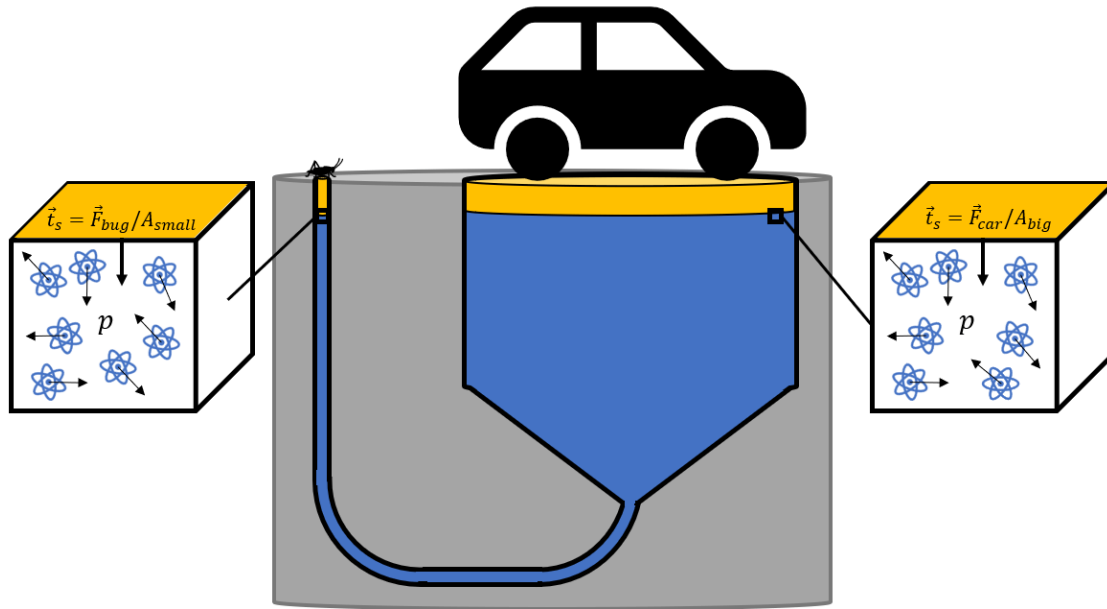


Figure 02.2: A bug and a car are placed on two ends of a hydraulic jack. The bug's weight induces a force per unit area on the fluid, increasing its pressure. This pressure increase multiplied by the surface area of the large piston is enough to balance the weight of the large car.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. Think of some simple examples of force fields and calculate/infer what the pressure distribution they induce in a still fluid might look like. Can you think of any force field that leads to nonsense pressure results? What would that mean?
3. Can you think of a way to determine when the incompressibility

assumption is correct? What properties would you need to know? Can you think of a "metric" to determine how correct the assumption is?

4. If you heat up a liquid in a kettle or frying pan, you would usually reduce its density quite noticeably. How does this not contradict the statement that external continuum forces don't affect fluid densities?
5. Solids can store energy in molecular bonds when pushed or pulled, behaving in a way similar to a tiny bunch of connected springs. How is this behavior different from the behavior of a fluid? How is it similar?
6. Why isn't the density of a fluid a function of the fluid velocity, especially if we stated well-defined densities occurs as a result of molecular motion?
7. Can you think of a way to mathematically derive that  $|\vec{t}_s| = p$ ? What would you need to consider?
8. What are the units of pressure, and it is a scalar or a vector? What are the units of traction, and is it a scalar or a vector? Can you think of reasons why people might confuse the two?

9. Can you think of some applications for the concept of hydraulics described above? Are there any you can identify that have already been made?

## Further Reading & Context

The dependence of density on pressure is a common topic in thermodynamics textbooks, particularly in stationary gases—a general introduction to the subject can be found in [62] among many others. The description (and justification) of incompressibility shown here is similar to that presented in [5]. Most textbooks define pressure *a priori* as a surface traction ([34][18][47][33][49][53]), as a result of the fact that pressure as a thermodynamic concept fails to connect to its mechanical counterpart in perfectly incompressible (and therefore unreal) fluids—see [3], [51] or [34] for a discussion of this. This is not a problem, as incompressibility is always regarded in this text as an approximation. An entropic construction of the notion of pressure can be found in [62]. Introductory treatments to the fundamentals of flow that is compressible can be found in [2], [49], [53], [72], and [42] among many others.

# 03

## BUOYANCY



*“The Great Wave off Kanagawa”, Katsushika Hokusai.*

With a decent understanding of hydrostatics and the way pressure and forces interact under our belt, we can then begin to ask some important questions about how objects react to the presence of fluids. For example, how do I know if a boat will sink or float? Is thinking about sinking and floating the only thing I need to thinking about when I'm designing a boat?

To be able to answer those questions, we need to recall the example we saw before of the fluid point at the bottom of the cup. There, we saw that a surface force  $\vec{f}_s$  of the same magnitude as the pressure within the fluid point ensures that the fluid point remained stationary. This fact turns out to generalize for any interface between two continuum objects, be they solid or fluid; at every point of the interface between two continua, there is a traction with magnitude equal to the pressure at that point, pointing perpendicular to the interface in the direction of the continua of interest. This is what we saw in the cup example—in that scenario, our continuum of interest was actually the water, and the cup was exerting a traction on the fluid point proportional to its local pressure  $|\vec{t}_s| = p$ .

This concept allows us to understand & calculate the hydrostatic forces acting on an object embedded in a fluid through the following analysis:

1. Determine the pressure distribution of the fluid.

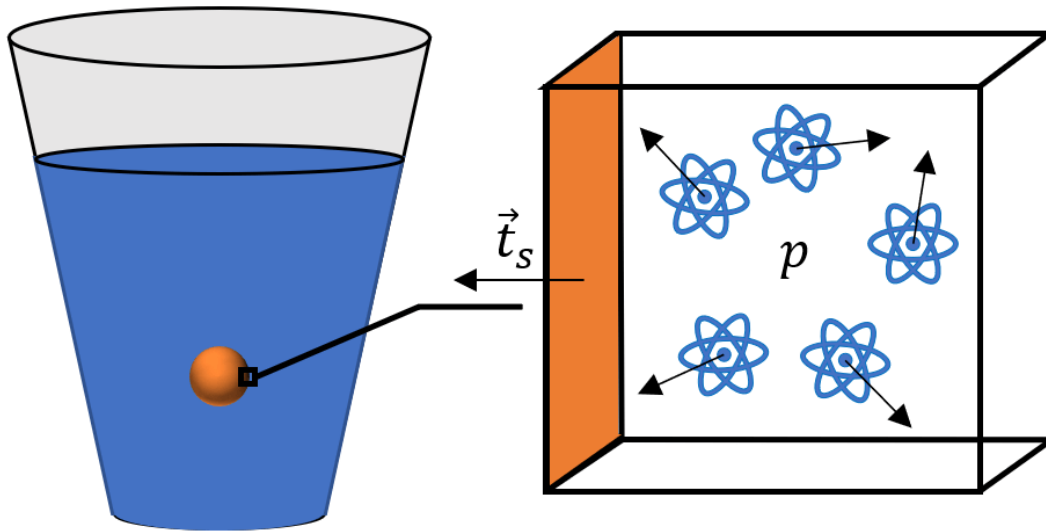


Figure 03.1: A solid sphere submerged in a cup of water experiences a traction at every point on its surface, which is pointing perpendicular to the surface at that point, and with magnitude equal to the fluid pressure at that point.

2. Determine the interface between the continuum of interest and the fluid.
3. Determine the pressure of the fluid at the interface.
4. Calculate the pressure-induced traction at each point on the interface.
5. "Add up" (integrate) those tractions to get the total hydrostatic force on the object.

We can express this last step of the process succinctly using mathematical language:

$$\vec{F}_b = \int_A \vec{t}_s dA$$



This procedure involves applying tools from vector calculus, which beginning practitioners of fluid mechanics might find daunting. Luckily, we can obtain the total buoyancy force on any object embedded in a static liquid on Earth without having to do so. In short, the buoyancy force on an object in a fluid is always equal to the weight of the fluid it displaces, and always points up. This is mathematically represented as Archimedes' law:

$$\vec{F}_b = -\rho V_{\text{displaced}} \vec{g}$$

For the sake of ideological consistency, we can show how Archimedes' simple law is derived from the more complicated vector calculus process of adding up tractions described above. This relies chiefly on the fact that the pressure distribution in a given static fluid on Earth is essentially always the same, stemming from the solution to the equation  $\nabla p = \rho \vec{g}$  we derived in Chapter II. Since the gravitational force points strictly downwards, and  $\vec{g}$  has a constant magnitude, we can simply integrate with respect to the "depth" direction  $z$  to find that the pressure field is

$$p = p_0 + \rho |\vec{g}| z$$

where  $p_0$  represents the pressure at the surface of the fluid and  $z$  represents the depth of the fluid at which the fluid point is located.

Clearly, the pressure of a fluid point here only depends on its depth within the fluid. As a result, horizontal pressure-induced

tractions that act on the surface of an object in such a fluid must cancel out, since a pressure-induced traction on the "left" side of the object will inevitably be canceled by traction on the "right" side with the same net magnitude.

Now consider a cube of side length  $a$  within the fluid. Its interface is determined by six distinct surfaces; the top, bottom, and four sides. Because the traction on the side surfaces must necessarily cancel by the argument above, the only contributions to the buoyancy force are going to come from the top and bottom, in which the tractions are uniform but distinct.

Therefore, the total traction on the cube is just the difference in tractions between the top and bottom of the cube, multiplied by the area over which they act:

$$\vec{F}_b = \int_A \vec{t}_s dA = a^2 \vec{t}_{\text{top}} + a^2 \vec{t}_{\text{bottom}}$$

$$\vec{F}_b \cdot \hat{y} = a^2(p_0 + \rho|\vec{g}|z_{\text{bottom}}) - a^2(p_0 + \rho|\vec{g}|z_{\text{top}})$$

$$\vec{F}_b \cdot \hat{y} = a^2 \rho |\vec{g}| (z_{\text{bottom}} - z_{\text{top}})$$

$$\vec{F}_b \cdot \hat{y} = a^3 \rho |\vec{g}| = V_{\text{cube}} \rho |\vec{g}|$$

Because this force is additive in the volume, and because every solid body can be approximated to arbitrary accuracy as a combination of sufficiently small cubes, we have by consequence derived Archimedes' law for arbitrarily shaped solid objects. Voilà!

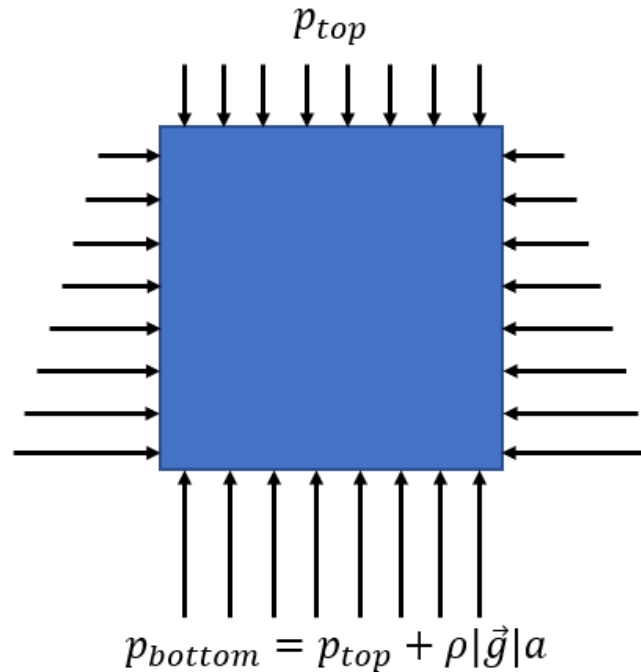


Figure 03.2: Frontal view of a submerged cube with side length  $a$  and the pressure-induced tractions it feels. From symmetry, the tractions on the sides are equal and opposite, and the net buoyancy force comes from the difference in tractions between the top and bottom of the cube.

This begs the question; why bother thinking about this in a way that requires vector calculus if we don't need it to calculate the buoyancy force? Luckily, the answer is simple—rotation!

Think of a cylinder wrapped in string. If you pull the string on both sides with equal and opposite force, the center of mass will surely remain stationary by virtue of Newton's law, but the cylinder as a whole will spin about its axis. Clearly, just the total force on an object doesn't paint the whole picture of how the cylinder moves; the location of those forces on the object also matters! And since we

don't want our boats to spontaneously capsize as we sail on the ocean, understanding this phenomenon is imperative for any practical application of buoyancy.

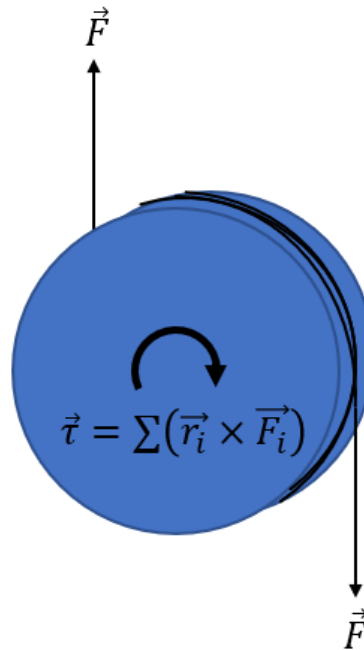


Figure 03.3: A cylinder wrapped in string is pulled on opposite ends by equal and opposite forces. Although the cylinder's center of mass doesn't move due to zero net force, the cylinder spins due to a torque equal to the sum of the moments induced by the forces.

In essence, hydrostatic pressure-induced tractions don't only induce a net buoyancy force  $\vec{F}_b$  on an immersed object, but a buoyancy torque  $\vec{\tau}_b$  as well. And unlike the buoyancy force, we don't really have a neat torque version of Archimedes' law that lets us calculate this buoyancy torque without using vector calculus. As a result, we find the process of calculating it nearly identical to the original process of

determining the net buoyancy force:

1. Determine the pressure distribution of the fluid.
2. Determine the interface between the continuum of interest and the fluid.
3. Determine the pressure of the fluid at the interface.
4. Calculate the pressure-induced traction torques ( $\vec{r} \times \vec{t}_s$ ) at each point on the interface.
5. "Add up" (integrate) those traction torques to get the total hydrostatic torque on the object.

We can again list out this last step succinctly using mathematical language:

$$\vec{\tau}_b = \int_A \vec{r}_{cm} \times \vec{t}_s dA$$

where  $\vec{r}_{cm}$  represents the position of the point being analyzed relative to the object's center of mass.

As it turns out, this process of summing up small contributions through integration is a critical tool in all areas of fluid mechanics, which we shall soon observe.

## Things to Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?

2. How would you determine if an object will sink or float? How would you determine if an object will spin when it either sinks or floats, and in which direction?
3. A static, floating object is always partially submerged in the liquid below it. Can you determine the volume of the submerged part of the object using the techniques above? (Ignore air, then ask yourself why you could do that.)
4. Think of the pressure-induced traction distribution on a triangle. Convince yourself that the horizontal tractions cancel out. Which way would the triangle spin if it floats? Is it the same direction if it sinks?
5. Use the result above to justify why boat hulls look the way they do. Try to think of reasons for the design of boat hulls in general (shape, length, width, depth, etcetra).
6. If an object possesses a uniform identical density to water, it will neither sink nor float—its center of mass simply remaining stationary. In this scenario, could it rotate due to hydrostatic forces?

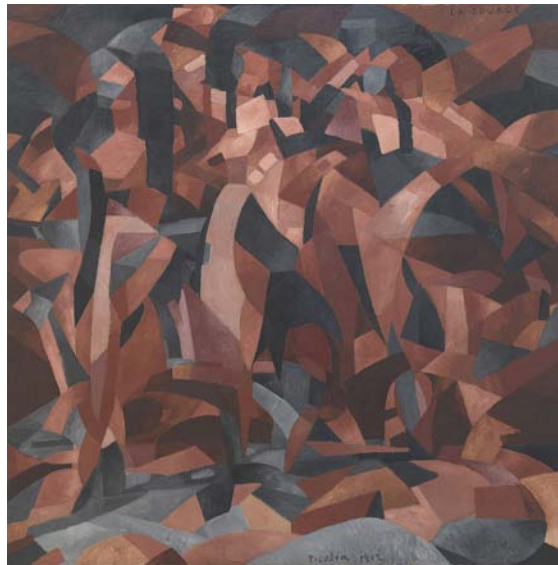
## Further Reading & Context

The treatment of buoyancy forces and derivation of Archimedes' law discussed here is essentially equivalent to that in most introductory textbooks that speak on the subject ([33][5][30][21][2]). Discussions on buoyancy torques are far more scarce, but are discussed in the context of submerged object stability in [30] and [21] in greater detail than in this text. The reader should be aware that some fluid mechanical textbooks do not feature any descriptive content on buoyancy and Archimedes' principle ([34][38][3]).



# O4

## CONTROL VOLUME ANALYSIS



*"The Source", Francis Picabia.*

As we showed previously, the idea of adding up small contributions on a surface is a powerful concept in fluid mechanics. As a matter of fact, studying the way that moving fluids apply forces to objects also relies on this principle, and we can promptly use the techniques we saw utilized for buoyancy to this purpose.

To do so, consider an imaginary surface enclosing some amount of unmoving fluid, through which fluid could potentially move in and out of—very similar conceptually to the "net" we discussed in Chapter 1. This surface, and the volume it encloses, is commonly referred to as a control volume. Just like a solid object submerged in a fluid, the fluid inside of the surface feels a net force as a result of the contributions from the pressure-induced tractions on the imaginary surface bounding it. In addition, the fluid can also feel net forces due to force densities distributed inside of it, like gravity. Stating this mathematically,

$$\vec{F} = \int_V \vec{b} dV + \int_A \vec{t} dA$$

where  $\vec{b}$  represents the force densities within the fluid.

Let's take a look at a quick example of using this equation by analyzing static fluid in a pipe floating in space. Let's say that the pressure at one end of our control volume in the pipe is  $p_1$ , and the pressure at the other end is  $p_2$ . Since there isn't any gravity to worry about (thanks space!), the only forces acting on the fluid are the

pressure-induced tractions acting on the surface of the control volume. If these pressures are different, a net force acts on the fluid inside the control volume—causing it to flow. As you might have expected, we’ve proved something that is usually intuitive for most; differences in pressure within a fluid, in the absence of countering forces, generate fluid flow.

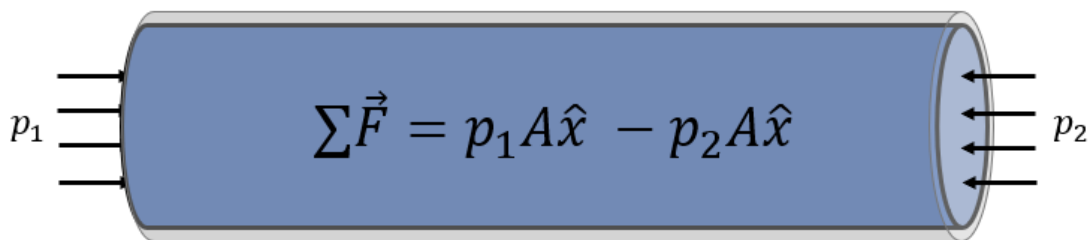


Figure 04.1: Pressure-induced tractions act on a volume of fluid in a pipe, which is described by a control volume. If the pressures are not equal, the fluid within the control volume experiences a net force, causing it to flow.

But forces aren’t the only thing we can analyze with this trick. In fact, we can use this technique to analyze the change of essentially any net property within a control volume!

For example, take into consideration another control volume surrounding a pipe, again in space, filled with a moving fluid containing Chemical X. How would you determine if the amount of Chemical X within the control volume is accumulating (or decreasing) over time, and by how much? Well, there are only two possible ways through

which the chemical can enter/exit the control volume; if it was somehow being spontaneously created within the pipe, and by moving into or out of the pipe section. Writing this down mathematically, you'd get the following equation for the rate of change of the mass of chemical X in the control volume over time:

$$\frac{\partial m_X}{\partial t} = \dot{m}_{X\text{generated}} + \dot{m}_{X\text{moving in}} - \dot{m}_{X\text{moving out}}$$

where the dots indicate a rate of change over time. This equation is fairly useless in its current form, but coming up with specific forms for each term will lead to some useful expressions.

Because Chemical X might not be generated in a uniform way within the control volume, the first term in the equation above should actually be the sum of a bunch of tiny contributions within the control volume, each generating (or destroying) some small amount of Chemical X at each fluid point. In other words, the net amount of Chemical X generated or destroyed in the control volume over time comes from adding up changes in the density of Chemical X at each point inside of the fluid volume. Such a contribution is usually called a generation, reaction, or more generally a source, in allusion to the fact that they represent sources (or sinks) of production of a quantity.

Regarding the other two terms, which determine the rate of movement of Chemical X into the control volume, we again find that they come from adding up small contributions of Chemical X, this time

moving into or out of points on the surface of the control volume. These small contributions are usually referred to as fluxes (or occasionally flux densities), and are usually mathematically represented with the symbol  $\vec{j}$ .

To calculate these, we first need to determine the amount of moving Chemical X at a given point, which occurs as a result of two distinct physical processes. The first is advective flux, which is due to Chemical X flowing into (or out of) a point along with the fluid it's immersed in; this is just the density  $\rho_X$  of Chemical X at the point, multiplied by the velocity of the fluid at that point  $\vec{v}$ . The second is diffusive flux, which is a type of motion we haven't discussed previously, which is when molecules of Chemical X spread out from random motion to eliminate gradients in the density of Chemical X, independently from the fluid's flow. This flux is calculated by multiplying the negative gradient of Chemical X at a point,  $-\nabla\rho_X$ , with a constant  $D_X$  commonly referred to as a diffusivity.

But to determine how much of that movement is going in or out of the control volume, we need to add some kind of extra quantifier that makes the flux contribution zero if the movement is parallel to the surface, positive if the movement is going into the volume, and negative if the movement is going out of the volume. Mathematically, we do this by taking the velocity vector at a point on the surface and dot-producting it with a unit normal vector, which always has a

magnitude of 1 (hence unit) and is defined to be always pointing out of the surface of the control volume (hence normal). Doing this has the nifty benefit that we don't have to worry about distinguishing which terms represent movement into the control volume and which represent flow out; the dot product takes care of that for us.

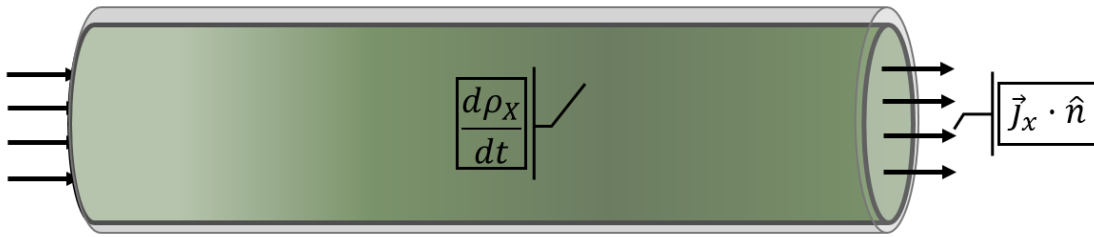


Figure 04.2: A control volume is generated in a section of pipe containing fluid with Chemical X. The total amount of Chemical X in the pipe changes only as a result of two things; the creation/destruction of Chemical X within the pipe (sources/sinks), and the movement of Chemical X into/out of the pipe (fluxes).

Finally, we note that the total mass of Chemical X inside the control volume can be calculated by summing up all the density contributions within the control volume. With this in mind, and combining all of these expressions, we get the following equation:

$$\frac{\partial}{\partial t} \int_V \rho_X dV = \int_V \frac{d\rho_X}{dt} dV - \int_A (\vec{j}_X \cdot \hat{n}) dA$$

$$\frac{\partial}{\partial t} \int_V \rho_X dV = \int_V \frac{d\rho_X}{dt} dV - \int_A \rho_X (\vec{v}_X \cdot \hat{n}) dA + \int_A D_X (\nabla \rho_X \cdot \hat{n}) dA$$

This first equation is commonly referred to as the Reynolds transport theorem, and it is incredibly general—so let's put this equation's usefulness to the test! Consider a pipe in space again, but this time filled with moving water, and with inlets and outlets of different cross-sectional surface. Let's say we've been running water through this pipe for a relatively long time, so nothing within the pipe is changing. Let's also assume that water is incompressible, so the density of water is the same everywhere. Can we make any statements about the speed of water coming out of the pipe relative to the speed of water coming in?

We can try to do so by performing a control volume analysis around the pipe. Doing so, we find the following:

1. The mass of water inside the pipe isn't changing, since we've been running it for a long time:  $\frac{\partial}{\partial t} \int_V \rho_X dV = 0$
2. Water isn't being generated or destroyed in the pipe through a chemical reaction or anything like that:  $\int_V \frac{d\rho_X}{dt} dV = 0$
3. The velocity of the water is always pointing either directly into or out of the pipe:  $\vec{v}_{\text{inlet}} \cdot \hat{n} = -v_1, \vec{v}_{\text{outlet}} \cdot \hat{n} = v_2$
4. The water is incompressible, so the water's density at the inlet and outlet are the same and there aren't any density gradients anywhere:  $\rho_{\text{inlet}} = \rho_{\text{outlet}} = \rho, \nabla \rho = 0$

Plugging all of this in, we find:

$$\frac{\partial}{\partial t} \int_V \rho \, dV = \int_V \frac{d\rho}{dt} \, dV - \int_A (\vec{j} \cdot \hat{n}) \, dA$$

$$0 = - \int_A \rho (\vec{v} \cdot \hat{n}) \, dA$$

$$0 = - \int_{\text{inlet}} \rho_{\text{inlet}} (\vec{v}_{\text{inlet}} \cdot \hat{n}) \, dA - \int_{\text{outlet}} \rho_{\text{outlet}} (\vec{v}_{\text{outlet}} \cdot \hat{n}) \, dA$$

$$0 = \rho \left( \int_{\text{inlet}} v_1 \, dA - \int_{\text{outlet}} v_2 \, dA \right)$$

$$\boxed{\int_{\text{inlet}} v_1 \, dA = \int_{\text{outlet}} v_2 \, dA}$$

Take a look at that; it turns out that the integral of the water's speed over the inlet has to match the speed integral over the outlet! Roughly speaking, that means that when you have a pipe with steadily flowing water whose inlet is larger than its outlet, the average water speed at the outlet will be bigger than at the inlet. This is why putting your thumb over your sink's faucet will cause the water to shoot out quickly! A device that exploits this speed-changing phenomenon in engineering is called a nozzle, and you can find one in just about anything that involves fluids flowing inside of something.



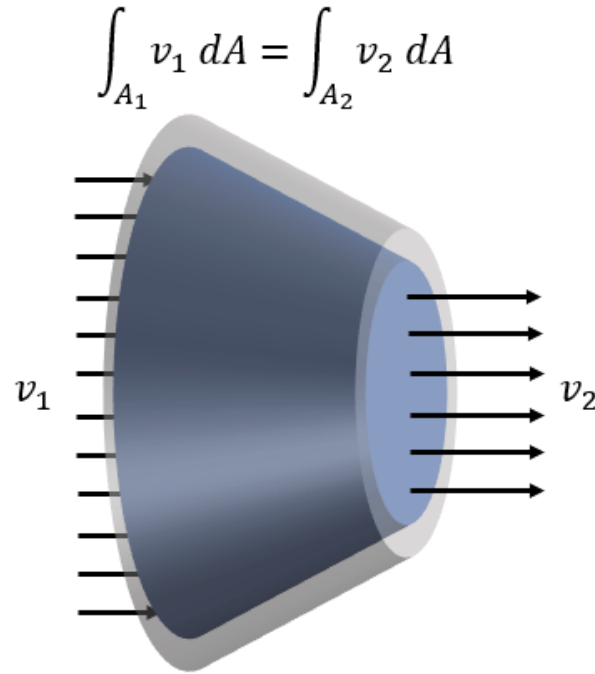


Figure 04.3: In order for fluid to not accumulate within a nozzle, the integrals of speed over the area of the inlet/outlet need to match, which causes fluid at the nozzle outlet to be generally faster than fluid at the nozzle inlet.

Let's conclude by considering what happens if we use the concepts above to construct an equation for the rate of change of the momentum  $\vec{P}$  of the fluid inside a control volume:

1. The net momentum inside the control volume  $\vec{P}$  is the sum of contributions of momentum density  $\rho\vec{v}$  within it, and so the rate of change of momentum in the control volume is equal to the rate of change of those contributions:  $\frac{\partial \vec{P}}{\partial t} = \int_V \frac{\partial (\rho\vec{v})}{\partial t} dV$
2. Momentum is generated or destroyed inside of the control

volume only as a result of body forces within the control volume, per Newton's second law:  $\int_V \frac{d(\rho \vec{v})}{dt} dV = \int_V \vec{b} dV$

3. Momentum moves into the control volume through momentum density fluxes on points on the surface, either through directly imposed tractions on the surface, or momentum density flowing & diffusing in:  $-\int_A \rho \vec{v} (\vec{v} \cdot \hat{n}) dA - \int_A (\vec{j}_{\text{diff}}^{[\rho \vec{v}]} \cdot \hat{n}) dA + \int_A \vec{t} dA$ . I'm keeping the explicit mathematical form of the diffusive momentum density flux term,  $(\vec{j}_{\text{diff}}^{[\rho \vec{v}]} \cdot \hat{n})$ , hidden for now. All you need to know now is what the term represents.

Putting everything together, we find an equation that tells us how the momentum of a control volume changes under the effects of external forces and fluid flow:

$$\frac{\partial \vec{P}}{\partial t} = \int_V \vec{b} dV + \int_A \vec{t} dA - \int_A (\vec{j}_{\text{diff}}^{[\rho \vec{v}]} \cdot \hat{n}) dA - \int_A \rho \vec{v} (\vec{v} \cdot \hat{n}) dA$$

That last term is indicating to us that fluid flow can induce forces! It's also telling us that fluid flow can induce changes in the momentum of the control volume that contains it—to see this in action, just consider some kind of device (in space again, to ignore gravity) that is shooting out incompressible fluid at some constant velocity  $\vec{v}$  directly behind it. Making a control volume analysis around the device, we find that the change of the momentum of the object has to be  $\frac{\partial \vec{P}}{\partial t} = -\int_A \rho \vec{v} |\vec{v}| dA$ , which pushes the device in the opposite direction of

the fluid's speed. Using a fluid to induce movement in this way is commonly referred to as jet propulsion, and the device itself can be referred to as a jet (or more arguably, a rocket).

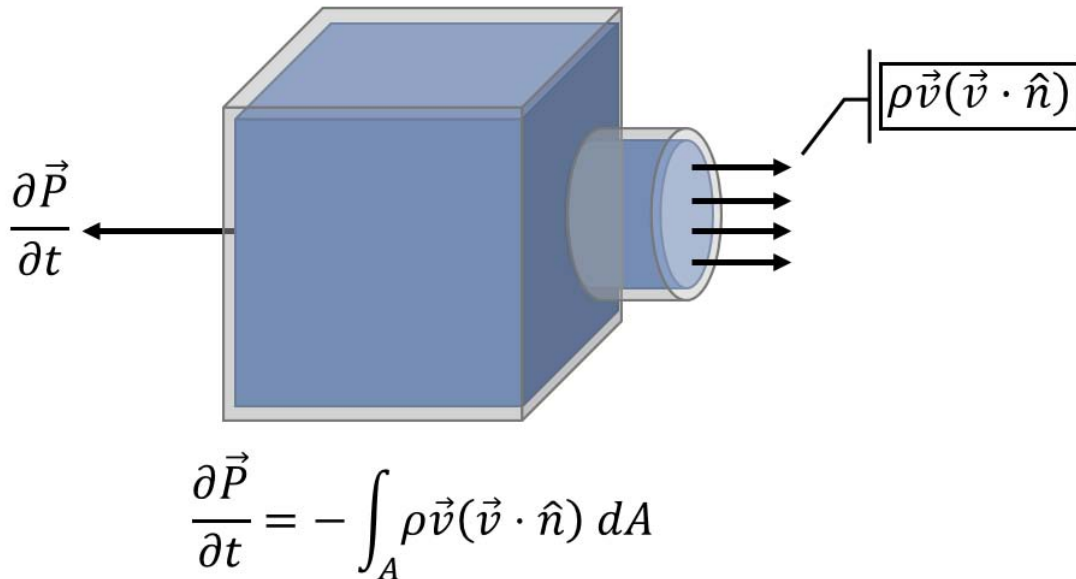


Figure 04.4: A jet/rocket shoots out fluid behind it. Because the fluid is carrying momentum away from the control volume, the fluid in the control volume experiences a net force in the opposite direction, pushing the jet/rocket forward.

Harkening back to our conclusion in Chapter 1, the control volume analysis we've performed here is letting us understand the different ways pressure, velocity, and density affect each other in a variety of different specific contexts. And to verify its utility, we should spend some time applying this analysis towards the principal engineering application of fluid mechanics; hydraulics.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. What happens when you include gravity in the examples? Does anything change?
3. Why is the diffusive flux proportional to the negative of the gradient of the diffusing quantity?
4. Say you wanted to design a jet/rocket, and were trying to maximize the amount of force your jet/rocket experiences. Does it benefit you to use a nozzle? If so, what kind? Try to figure it out with control volume analysis.
5. Could you use that last equation to get an estimate for the kind of motion a pressure difference across a pipe induces? What would you need to consider?
6. Can you try to use the techniques above to derive an equation for the net torque on a fluid in a control volume? What would each term represent?

7. Let's say you shoot a stream of water horizontally onto a wall, and the wall deflects the water straight up and down in two identical streams. How much force is the wall experiencing?

## Further Reading & Context

Control volumes are ubiquitous in the fields of fluid mechanics and heat transfer, and can be found discussed in [33], [21], [3], [49], [53], [34] and [72] among many others. Similarly, the Reynolds transport theorem is discussed in many fluids textbooks ([3][33][21][3][49][72], [53][3], etc.) in the context of deriving the equations of fluid dynamics, although some books ([47][37][38]) refer to it indirectly as "differentiation under the integral sign". This textbook deviates from many others in that it does not discuss the notion of Eulerian/Lagrangian formulations or the idea of a material derivative—called differentiation following the fluid in [47][37][5]—which I avoid to minimize mathematical confusion. Discussions of these can be found in [33], [34], [21], and [3], as well as the books listed immediately above.

The phenomenon of diffusion is only briefly mentioned in this text, and not discussed in any detail—the reader is encouraged to see [16], [7] or [54] for a comprehensive description of the role of diffusive fluxes in fluid dynamics and related fields, particularly chemical engineering.

# 05

## HYDRAULICS



*"Aqueduct Near Rome", Thomas Cole.*

The concept of control volume analysis we developed to analyze the way pressure, velocity, and density interact in fluids is surprisingly powerful. And in fact, such an analysis—coupled with a little bit of experimental data—forms the backbone of nearly all engineering applications of fluid mechanics. The simplest example of this, and certainly the most ubiquitous, is in hydraulics.

We had briefly seen hydraulics at the end of Chapter 2, where things like hydraulic jacks exploited the relationship of external forces and pressure. However, the study of hydraulics is far broader than that; in a nutshell, hydraulics is the field of engineering that exploits or manipulates enclosed fluids. The critical distinction between then and now is that we were exclusively looking at fluids that weren't moving, where the only thing we could manipulate was the pressure. Now that we have the machinery of control volume analysis, we can explore the full gamut of hydraulic phenomena that involve both pressure and velocity.

Let's consider a section of pipe again, this time filled with a moving incompressible fluid (so the density is constant everywhere). If we perform a control volume analysis for the mass of fluid inside the pipe section, and assume that the pipe flow has been running sufficiently long such that the total fluid mass in the pipe doesn't change, we find as we did in Chapter 4 that  $\int_{\text{inlet}} v_1 dA = \int_{\text{outlet}} v_2 dA$ . And although we don't know how the velocity changes at each point of

the inlet/outlet—we don’t have the mathematical machinery for that yet—we can define some average speeds at the inlet/outlet of the pipe  $\langle v_i \rangle$  such that those integrals turn into simple multiplications, with the assumption that all the flow at the inlet/outlet is perpendicular to the pipe cross-section. With that, we find the following equation:

$$\boxed{\langle v_1 \rangle A_1 = \langle v_2 \rangle A_2}$$

Although this is just a simplified and specific version of the conservation of mass equation we derived in Chapter 4, it is also what I like to call the first fundamental equation of hydraulics. It is fundamental because it is giving us a very simple and straightforward statement about the behavior of fluids that is very useful for engineering applications: the change of fluid speed between two ends of a steady hydraulic system involving incompressible fluids depends only on the geometry of that system.

This is relatively shocking, since one would certainly expect that pressure or gravity or external forces would play some kind of role in speeding up or slowing down the fluid. The fact of the matter is that they do, but only while the system is in the process of becoming steady; once flow is constant, the geometry determines flow speeds. The imperative thing to do now is to determine whether or not such a steady flow is attainable in a given system, and for that we need to figure out what’s happening with the pressure.



To that end, if we perform a control volume analysis for the momentum of the fluid in that same pipe, we find in a general sense:

$$\frac{\partial \vec{P}}{\partial t} = \int_V \vec{b} dV + \int_A \vec{t} dA - \int_A \left( \vec{j}_{\text{diff}}^{[\rho \vec{v}]} \cdot \hat{n} \right) dA - \int_A \rho \vec{v} (\vec{v} \cdot \hat{n}) dA$$

We can be far more specific now than we were in Chapter 4. However, we are going to consider an extra effect we didn't consider before; a momentum loss term  $\vec{H}$ . This momentum loss term is going to be extremely general, and is essentially accounting for all the phenomena that we don't have the tools to understand yet; friction, turbulence, recirculation zones, etcetra. With that stated, we make the following assumptions:

- Tractions on the inlet/outlet are pressure-induced.
- The only body force on the pipe section is gravity.
- The system has been in motion for a long time, so everything in the pipe section is in equilibrium ( $\frac{\partial \vec{P}}{\partial t} = 0$ ).
- The pipe section is relatively straight, such that defining a flow direction  $\hat{x}$  through the pipe makes sense.
- All forces perpendicular to the pipe inlet/outlet, and all tractions acting on pipe surfaces that aren't the pipe inlet/outlet, are balanced out by forces from the pipe itself (i.e. the solid structure surrounding the fluid).

- Diffusive momentum fluxes at the inlet/outlet are meager in comparison to the advective fluxes.
- The momentum loss term is always opposite to the flow direction, and always represents a loss ( $\vec{H} \cdot \hat{x} < 0$ )

This makes our expression a bit cleaner when we take the dot product with the flow direction  $\hat{x}$ :

$$0 = -|\vec{H}| + \int_V \rho (\vec{g} \cdot \hat{x}) dV - \int_A p \hat{n} \cdot \hat{x} dA - \int_A \rho \vec{v} (\vec{v} \cdot \hat{n}) \cdot \hat{x} dA$$

Defining average quantities just like we did before, we can swap out those integrals with products:

$$0 = -|\vec{H}| + \rho \vec{g} \cdot \hat{x} V + \langle p_1 \rangle A_1 - \langle p_2 \rangle A_2 + \rho \langle v_1 \rangle^2 A_1 - \rho \langle v_2 \rangle^2 A_2$$

To simplify the gravitational term, the volume  $V$  of the pipe section can be reasonably approximated as the average cross-sectional area of the inlet/outlet times the pipe length. Combining that with the dot-product term and doing some behind-the-scenes algebra, we can equivalently approximate the gravitational term as:

$$\rho (\vec{g} \cdot \hat{x}) V \approx \rho |\vec{g}| \frac{A_1 + A_2}{2} L \cos \theta = \rho |\vec{g}| \frac{A_1 + A_2}{2} (z_1 - z_2)$$

where each  $z$  represents the height of the center of the inlet/outlet relative to some fixed point, like the ground.

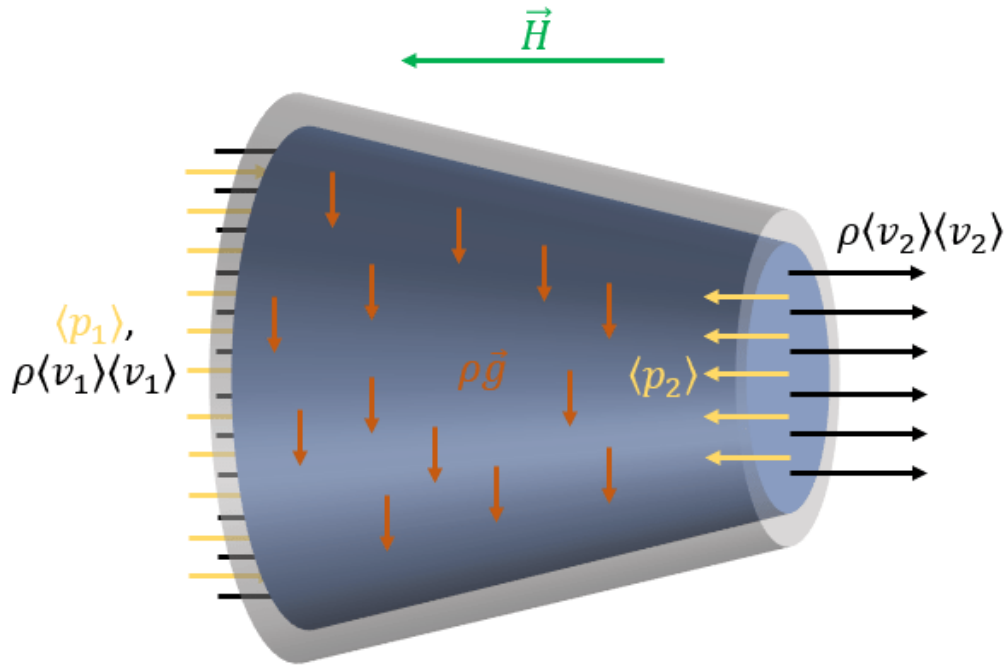


Figure 05.1: A control volume analysis over a hydraulic pipe section; each distinct contribution is in a different color. Gravitational body forces are in brown, pressure-induced tractions are in yellow, advective momentum fluxes are in black. The momentum loss, in green, has an unknown distribution within the fluid section but always has a net direction opposite to the flow.

Inserting back, and moving the momentum loss term to the left-hand side, we find the following equation:

$$|\vec{H}| = \rho |\vec{g}| \frac{A_1 + A_2}{2} (z_1 - z_2) + \langle p_1 \rangle A_1 - \langle p_2 \rangle A_2 + \rho \langle v_1 \rangle^2 A_1 - \rho \langle v_2 \rangle^2 A_2$$

I usually refer to this equation, which came from a control volume analysis for the momentum of a pipe section of fluid in the direction of flow, as the second fundamental equation of hydraulics. This is

because it is explicitly giving us the remaining pieces of the puzzle of understanding how the pressure changes in a pipe section; we know the changes in the velocity can be found entirely from geometric arguments thanks to the first fundamental equation, and everything coming from the gravitational term is geometric as well since it only depends on pipe section heights and cross-sectional areas. The only other thing we'd need to figure out the average pressure at the outlet, given some average pressure and velocity at the inlet, is that momentum loss term.

$$|\vec{H}| = \underbrace{\rho |\vec{g}| \frac{A_1 + A_2}{2} (z_1 - z_2)}_{\substack{\text{Considers friction,} \\ \text{turbulence,} \\ \text{recirculation, etc.}}} + \underbrace{\langle p_1 \rangle A_1 - \langle p_2 \rangle A_2}_{\substack{\text{Purely} \\ \text{geometric}}} + \underbrace{\rho \langle v_1 \rangle^2 A_1 - \rho \langle v_2 \rangle^2 A_2}_{\substack{\text{Geometric} \\ \text{after applying} \\ \text{first} \\ \text{fundamental} \\ \text{equation}}}$$

Figure 05.2: The second fundamental equation of hydraulics. Although it appears there are many unknowns, the only variables that remain unknown after applying the first fundamental equation are the pressures, one of which is usually already known, and the momentum loss term.

The momentum loss term  $|\vec{H}|$  is, as we stated before, extremely general; it is essentially accounting for every physical phenomenon associated with momentum loss in the fluid that we don't know how to describe, and is consequently a function of the pressures, velocities, and geometries of the pipe section in question. Luckily, decades upon decades of experimental work have empirically determined

accurate values for  $|\vec{H}|$  associated with specific phenomena and pipe geometries, such that hydraulicists only need to look up the values of  $|\vec{H}|$  for some given geometry, average flow pressure, and average flow speed. This momentum loss  $|\vec{H}|$  is almost universally approximated as proportional to the cross-sectional area, in which case hydraulicists define a momentum loss per area  $h = \frac{|\vec{H}|}{\langle A \rangle}$ , commonly referred to as hydraulic loss. This hydraulic loss  $h$  is usually further split into two confusingly named components; major hydraulic loss  $h_M$ , and minor hydraulic loss  $h_m$ . Major hydraulic loss should actually be called frictional loss or length-proportional hydraulic loss, as it is defined to be proportional to the pipe length and stems largely from friction, and minor hydraulic loss should be called geometric or length-independent hydraulic loss, as it accounts for geometrically-induced flow oddities like recirculation zones in pipe bends that are not proportional to pipe length. We can then rewrite the second fundamental equation of hydraulics as:

$$\frac{A_1 + A_2}{2} (h_M + h_m) = \rho |\vec{g}| \frac{A_1 + A_2}{2} (z_1 - z_2) + \langle p_1 \rangle A_1 - \langle p_2 \rangle A_2 + \rho \langle v_1 \rangle^2 A_1 - \rho \langle v_2 \rangle^2 A_2$$

We can do a couple of rapid-fire qualitative observations from these fundamental equations given some simple assumptions:

- If losses are negligible, and the pipe inlet/outlet are at the same height but the outlet is smaller than the inlet, the pressure at

the outlet has to drop relative to the inlet. If the outlet is bigger, the pressure has to increase. This is called the Venturi effect.

- If there is no flow, the second fundamental equation becomes an approximation of the integrated form of the hydrostatic equation.
- If losses are negligible, and the inlet/outlet areas are all the same, one can divide by the area to obtain something called Bernoulli's equation. I don't like that equation very much nor do I think it's useful, but it's important to mention it so that you know what other people mean when they reference it.

So, how do engineers apply these equations to an aqueduct or a network of pipes? Although there are many ways to do this, the common scenario is to consider some sort of pump or reservoir at a high and fixed pressure  $p_{in}$  and some outlet at a fixed lower pressure  $p_{out}$ , most of the time at atmospheric pressure. Then you guess a "reasonable" value for the velocity at that reservoir  $\langle v_{in} \rangle$ , use the fundamental equations of hydraulics to determine the inlet pressure and velocity of the next pipe section, and so on until you reach the outlet. Then, you see whether or not the outlet pressure you wind up obtaining with that trial guess of the reservoir velocity  $\langle v_{in} \rangle$  matches the correct outlet pressure; if it matches, great, if it doesn't, you keep trying with different reservoir velocities. If you can't find a match

at all, then steady flow in the system is likely impossible, and you need to tweak your hydraulic system. This sort of thing is called hydraulic circuit analysis or pipe network analysis, depending on the engineer's background.

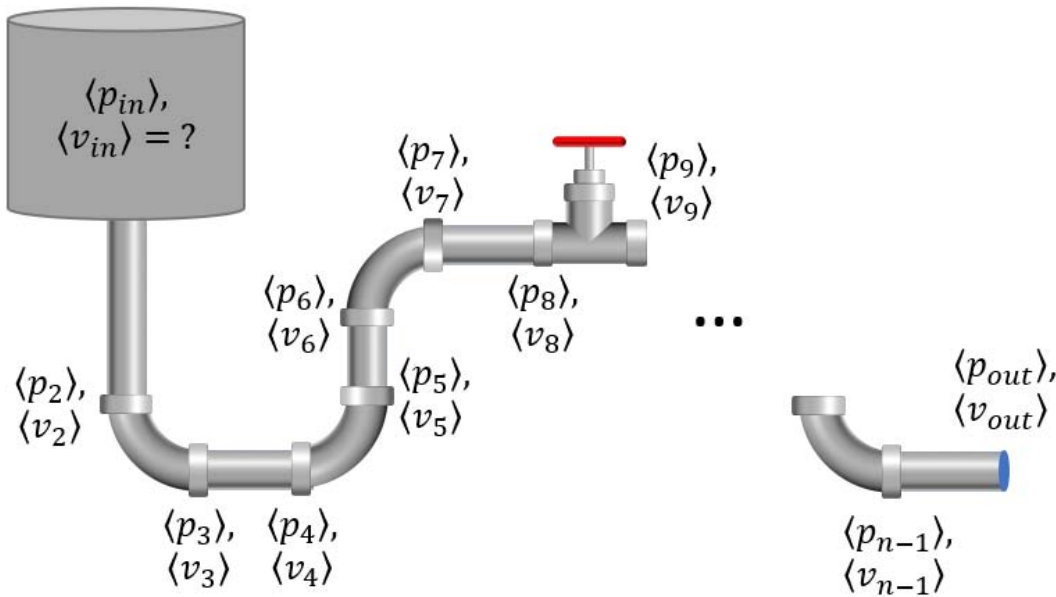


Figure 05.3: A hydraulic circuit or pipe network. Usually, one knows the reservoir/pump pressure and outlet pressure. Given a guess of the reservoir velocity, the fundamental equations of hydraulics are applied over every segment, starting from the reservoir, until the calculated pressure at the outlet matches the known pressure. Momentum losses for each segment have usually been previously determined by experiment for a given pressure/velocity change.

Using control volume analysis for the mass and momentum of fluid in a section of pipe at steady-state, along with some experimental data about momentum losses in pipes, we were able to get everything we needed to design and understand basic hydraulic sys-

tems. If we want to get a fundamental understanding of what those losses are and how they occur, though—if we want to get a fundamental understanding of the laws of fluid mechanics—we need to give our theoretical machinery of control volume analysis a little upgrade. That upgrade is called transport theory.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. Why do you think this kind of approach would be tricky when we're thinking about flow past something (like say, a plane)?
3. What details about the flow that we haven't found out yet would help us make this analysis even more specific and precise?
4. What processes do you think contribute to momentum loss in a curved pipe? What do you think they depend on?
5. Let's say you are trying to pump water into the 6th story of a building using just a pump, a curved piece of pipe, and a piece of pipe going straight up. How would you design the pump and the pipe geometry?
6. What do you think momentum loss due to friction depends on?



## Further Reading & Context

An introduction to applied hydraulics, with insight into the effects of valves/pipe elbows/etc., is provided in [30], [21], [49], [72], [53], and [33]. Many books, particularly those with a theoretical bent or those focused for a chemical engineering audience, ignore the subject of applied hydraulics ([34][47][37][38][7]). For "major" pressure losses in pipes, the central empirical tool of use is the Colebrook-White equation, first detailed in [12] and adapted into a graphical format called a Moody chart in [48]. For "minor" pressure losses, the practicing hydraulicist might rapidly find that the empirical data associated with them is mostly manufacturer-supplied information specific to a given part.

Perhaps the most significant difference between this text and other introductory fluid mechanics texts is that I essentially do not discuss the Bernoulli equation, which is a prominent feature of essentially every introductory fluids textbook ([33][47][34][21][37][3][5][30], [70][53][49][72][7]) and is usually introduced at around this point in those textbooks. This is done because, in my opinion, the Bernoulli equation is a simple approximation that allows for students to become familiar with the concepts of fluid mechanics through procedural, accessible numerical exercises, but is often a finnick and inconsistent analytical tool when it comes to predicting and analyz-

ing real world systems and can trick students into overstating the effectiveness of the Bernoulli approximation to real-world hydraulic systems. It is claimed in [19] that John von Neumann once said the study of fluids in which the Bernoulli equation holds is the study of "dry water"—I largely agree.

06

# TRANSPORT THEORY



*“Transport of Forces”, Fernand Léger.*

The control volume analysis technique we saw and used previously is both astoundingly general and eminently pragmatic. But it has a shortcoming; namely, that we have to define a specific volume before we can make statements about the fluid inside of it. This means that all of the statements we can make about fluids using control volume analysis depend on the specific system we're analyzing. If we want to make universal statements about the way fluids behave, i.e. describe the physical laws of fluid mechanics, we need to overcome this obstacle. As it turns out, this is easier than it seems.

Let's briefly summarize what control volume analysis dictates about some quantity in a fluid volume; the rate of change of the total amount of that quantity inside a volume is equal to the total amount of that quantity being generated or destroyed within it, plus the amount of that quantity which is moving in/out of the volume through its boundary surface. If we refer to the quantity we are interested in as  $\Xi$ , and its associated density as  $\xi$ , we can generate the following equation for the rate of change of  $\Xi$ :

$$\frac{\partial}{\partial t} \int_V \xi \, dV = \int_V \frac{d\xi}{dt} \, dV - \int_A (\vec{j}_\xi \cdot \hat{n}) \, dA$$

There's a neat mathematical trick we can do to turn that last surface integral into a volume integral called the divergence theorem. The details don't matter too much as long as you trust me; the only important thing is that the term still represents what it always did, flow

in/out of the control volume. Applying this trick, we find:

$$\frac{\partial}{\partial t} \int_V \xi \, dV = \int_V \frac{d\xi}{dt} \, dV - \int_V \nabla \cdot \vec{j}_\xi \, dV$$

Take a look at what we've managed to get; our equation, which represents the change in the total amount of some arbitrary fluid quantity, is exclusively in terms of adding up tiny contributions of different things within the control volume. So if we shrink the control volume down to a point, we don't have to bother with the integrations—we can look at an equation that relates those contributions directly! Remember, the concept of the equation is the same, we're just now using it to look at the way quantities accumulate within points rather than volumes.

To save ourselves considerable confusion from mathematical notation, I'm going to do what nearly every scientist does and relabel the  $\frac{d\xi}{dt}$  term to something simpler, like  $r$ ; it still represents the same thing, which is the generation/destruction of  $\Xi$  through some physical process going on in that point. Performing this relabeling, and shrinking things down to a point, we obtain:

$$\frac{\partial \xi}{\partial t} = r - \nabla \cdot \vec{j}_\xi$$

This type of equation is called a transport equation, and the process of using this type of equation to understand something is called a transport theory. Sometimes physicists call this kind of equation

a continuity equation, which is in my opinion silly and confusing. Moving the flux term to the other side of the equation, we get its final version:

$$\frac{\partial \xi}{\partial t} + \nabla \cdot \vec{j}_\xi = r$$

This equation is the most important equation in science. I am biased, sure, but you can model nearly everything with some version of it: fluid and solid mechanics, chemical kinetics, heat transfer, most of electromagnetism, general relativity, and a decent slice of quantum mechanics. Even statistical mechanics, if you play your cards right! This equation lets you mathematically understand the movement of anything through points in space, as long as you're able to understand the ways that your quantity of interest can move through space (its fluxes) and the ways your quantity is generated or destroyed at certain points (its sources and sinks).

Here's a simple and very useful example. If we wanted to look at setting up an equation for mass, you would first identify the corresponding quantity representing mass per unit volume (density  $\rho$ ) and write a transport equation for it down:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = r_\rho$$

Remember that all fluid mass movement is described by fluid velocity  $\vec{v}$ , so no need to include a diffusion term. In addition, in the absence of something goofy like nuclear reactions or relativistic phenomena,

we know mass can't be destroyed or created. That means that the  $r_\rho$  term representing the generation and destruction of mass has to be zero! Therefore, we get:

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

This equation describes the conservation of mass, and is the second-most important equation in fluid mechanics. If we additionally assumed that the liquid was incompressible, something interesting happens; the density can't change with respect to time or space, so any term proportional to that needs to vanish in the above equation. Expanding it out and getting rid of the density change terms, we get:

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

$$\nabla \cdot \vec{v} = 0$$

This equation refers to conservation of mass when a liquid is incompressible, and is sometimes called the incompressibility equation.

Even though mass in general is conserved, there might be some chemical reactions going on in our system of interest that change mass from one type of chemical to another. In that situation, you would set up transport equations for every chemical of interest in your system, and wind up with a system of equations for the chemical densities:

$$\frac{\partial \rho_i}{\partial t} + \nabla \cdot (\rho_i \vec{v}) - D_i \nabla^2 \rho_i = r_i$$

where the subscripts indicate a specific chemical in the system, and each source/sink term representing chemical reactions can depend on every other chemical density in the system. Such a system of equations is usually referred to as a reaction network. In chemical systems where there's no fluid flow, only diffusion and reactions occur, which are studied under the (apt) name of reaction-diffusion systems. Most chemical engineers ignore both the movement of fluid within the system and gradients in the densities of the chemicals, leaving only the reactions to drive the physical changes; this approximation is commonly called the CSTR approximation.

We can construct transport equations for energy as well; one very useful type of energy transport equation involves looking at the movement of thermal energy through a solid. The generic version of the equation should look something like this, according to transport theory:

$$\frac{\partial e}{\partial t} + \nabla \cdot \vec{j}_e = r_e$$

As it turns out, the heat energy density of a point is equal to the temperature  $T$  of the point, times the mass density  $\rho$  times the heat capacity  $c$ . In addition, because the solid doesn't flow, the only way energy moves through the solid is diffusively. As a result, we update the equation above to find:

$$\frac{\partial e}{\partial t} + \nabla \cdot [-D_e \nabla e] = r_e$$



$$\frac{\partial (c\rho T)}{\partial t} + \nabla \cdot [-D_e \nabla (c\rho T)] = r_e$$

Most thermal engineers like to assume the material properties of a solid that's transferring heat stay constant and are the same everywhere, so we can pull those properties (the heat capacity, density, and heat diffusivity) out of the gradients and simplify:

$$c\rho \frac{\partial T}{\partial t} - c\rho D_e \nabla \cdot \nabla T = r_e$$

$$\frac{\partial T}{\partial t} - \frac{D_e}{c\rho} (\nabla \cdot \nabla T) = \frac{r_e}{c\rho}$$

Cleaning up the vector calculus term, expressing  $\frac{D_e}{c\rho}$  as a single term  $\alpha$  representing the thermal diffusivity, and expressing  $\frac{r_e}{c\rho}$  as a single thermal generation term  $h$ , we find:

$$\frac{\partial T}{\partial t} - \alpha \nabla^2 T = h$$

Congrats! You just derived the heat equation using transport theory. Notice how many assumptions we had to make! And funnily enough, that heat diffusivity term  $D_e$  we saw above is almost always never called that; it's confusingly referred to as the thermal conductivity.

Hopefully this has convinced you of the generality and usefulness of transport theory. And with transport theory, we can now construct the most important equation in fluid mechanics; the transport equation for momentum.

If transport theory is to be believed, all I have to do is identify the momentum density, which is  $\rho\vec{v}$ , and write a transport equation for it that looks like this:

$$\frac{\partial(\rho\vec{v})}{\partial t} + \nabla \cdot \vec{j}_{\rho\vec{v}} = r_{\rho\vec{v}}$$

So far, so good. But if I go and write the advective flux term explicitly, what we find seems a bit confusing:

$$\frac{\partial(\rho\vec{v})}{\partial t} + \nabla \cdot \rho\vec{v}\vec{v} + \nabla \cdot \vec{j}_{\text{diff}}^{[\rho\vec{v}]} = r_{\rho\vec{v}}$$

How should we multiply these two vectors? It shouldn't be a dot product, because then we'd get a scalar, and we can't take the divergence of that. As it turns out, the right way to multiply these two vectors is with something called an outer product, which looks like this:  $\otimes$ . And the outer product of two vectors is a weird mathematical object called a tensor, which we will see a lot of in the next section.

Equipped with this knowledge, we rework the momentum transport equation to:

$$\frac{\partial(\rho\vec{v})}{\partial t} + \nabla \cdot (\rho\vec{v} \otimes \vec{v}) + \nabla \cdot \vec{j}_{\text{diff}}^{[\rho\vec{v}]} = r_{\rho\vec{v}}$$

It might make you uncomfortable to have that tensor expression up there; most students don't learn how to do calculus with tensors until after fluid mechanics (if they ever do). Luckily, we can break the term up into terms that can be handled entirely with vector calculus,

leading us to the Lamb form of the momentum transport equation:

$$\frac{\partial(\rho\vec{v})}{\partial t} + \frac{1}{2}\nabla(\rho\vec{v} \cdot \vec{v}) + (\nabla \times \rho\vec{v}) \times \vec{v} + \nabla \cdot \vec{j}_{\text{diff}}^{[\rho\vec{v}]} = r_{\rho\vec{v}}$$

This is helpful to understand what the tensor term in the equation means, but I personally don't find it helpful for keeping track of what the equation means; which is that it's a transport equation for momentum in a fluid. We'll revisit the Lamb form later, but right now I'd like to stick with the  $\nabla \cdot (\rho\vec{v} \otimes \vec{v})$  notation for conceptual clarity.

At this point, it seems tempting to simply say that the only source or sink of momentum in a fluid point comes from body forces that are external to the fluid. But we know that's not true! We know pressure gradients cause changes in momentum within a fluid, and the momentum for that is coming entirely from the "storage" of molecular energy within the fluid. In addition, we're missing another huge factor; friction. With friction, the opposite happens—energy in the movement of a macroscale fluid gets dissipated into microscale molecular energy of random motion. The common factor between pressure-induced forces and friction is that both of them are causing macroscale, fluid effects as a result of microscale molecular phenomena.

Noting that momentum density diffusion is also a molecular effect, I find it helpful to lump in all the molecular effects together

into a simple term  $\vec{r}_{\text{molecular}}$  and rewrite the momentum transport equation in the following way:

$$\frac{\partial(\rho\vec{v})}{\partial t} + \nabla \cdot (\rho\vec{v} \otimes \vec{v}) = \vec{f} + \vec{r}_{\text{molecular}}$$

Although it may not look like it yet, this is the most general form of the mathematical basis for essentially all of fluid mechanics: the Cauchy momentum equation.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. Let's say you're a nuclear engineer and have to deal with the unpleasant situation of handling a radioactive fluid that is routinely converting its mass into radioactive energy emissions. How would you write the mass transport equation for it?
3. How would you describe heat transport in a fluid? Is it more efficient in a solid, or less? Does this help you understand some common design features in thermal engineering?
4. Knowing what you know about the momentum transport equation, what do you think is the form of a transport equation for an arbitrary vector?

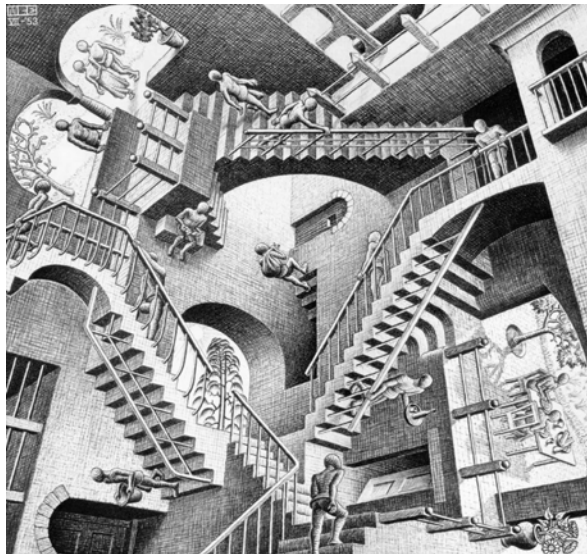
5. I never wrote out the momentum diffusive flux term. Knowing what you know about the generic mathematical form of diffusive fluxes, can you figure out why I chose to do that?

## Further Reading & Context

There are many textbooks focused on transport phenomena as a whole, with varying degrees of emphasis on fluid dynamics ([7][16][41]). The laws of solid mechanics are also formulated using transport theory; see [64] or [45] for a derivation of these. The laws of classical electromagnetism can be expressed in transport theory form equivalent to Maxwell's equations, although the transport form must (as far as I know) be derived from Maxwell's equations, not independently of them—see [22] for a description of this process. An accessible introduction to the use of transport models in electrohydrodynamic systems can be found in [32]. Formulations of quantum mechanics using transport theory which are equivalent to current wave function formulations, although unpopular due to their analytical intractability, exist—the pioneering example being [43]. See [9], [44] or [3] for a discussion of the divergence theorem, sometimes called Gauss's divergence theorem or Gauss's theorem, among other useful theorems in vector calculus.

07

## INTERMEZZO: TENSORS



*“Relativity”, M.C. Escher.*

Before we continue on with our study of fluid mechanics, it's best to make a quick pit stop and make sense of that strange concept I mentioned previously; that of a tensor, which we saw in the form of that strange multiplication  $\rho \vec{v} \otimes \vec{v}$ . The "proper" way of defining a tensor is hotly debated by mathematicians and physicists of different fields, so my idea here is not necessarily to rigorously define them, but to give you a brief sense of what a tensor is and how we use them in fluid mechanics. In my experience, that is best illustrated by revisiting the concept of a vector, like the velocity  $\vec{v}$ .

Knowing the velocity of an object tells us two things; the speed of the object, which is just some number like 20 miles per hour, and the direction in which it is traveling. But as anyone that's said "No, I meant my left" can attest to, direction is relative—whenever we talk about velocity in its most general sense, we usually describe it in terms of three independent components representing the speeds in specific, perpendicular directions that we've arbitrarily defined in the 3-dimensional space we live in (usually denoted with  $x, y, z$ ). Changing those arbitrarily defined reference directions must then necessarily change the expressions for the components, commonly referred to as projections, even though it doesn't physically change the velocity of the object itself.

That being said, velocity doesn't intrinsically possess three directions. As we stated above, it possesses just one, the direction in which

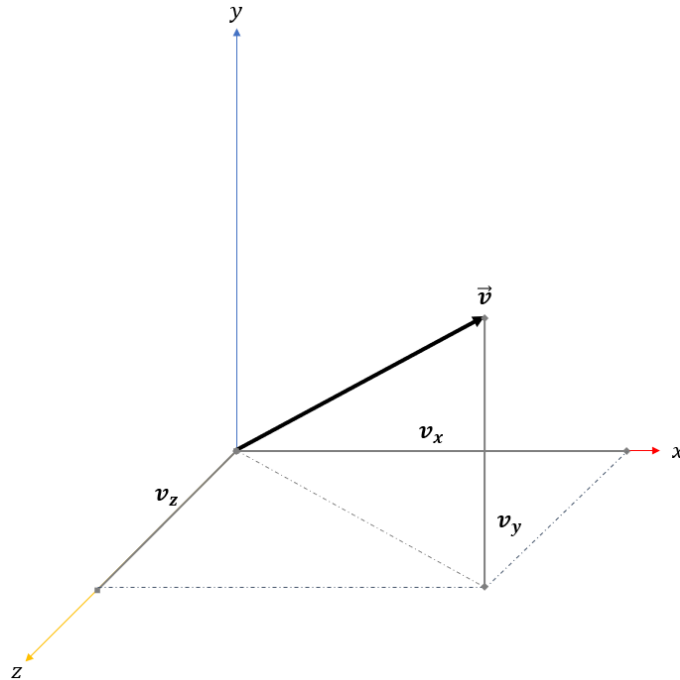


Figure 07.1: A velocity vector, in black, along with its three components (or projections) onto a set of three distinct directions represented by coordinate axes. Each component represents the speed of the object in the direction the component represents. If I spun the axes around, the components would change, but the vector itself wouldn't.

the object is instantly traveling in. With that in mind, a tensor is just a mathematical object that possesses any number of directions. People usually refer to the amount of directions a tensor possesses as its order, and so we can straightforwardly spot that the velocity is a first-order tensor because of its single "natural" direction.

If the velocity is a first-order tensor, then it might seem obvious to state that the mathematical object  $\rho \vec{v} \otimes \vec{v}$ , which we said represented the momentum density flux, possesses two directions, and



is as such a second-order tensor. In this case, those directions are straightforward to spot; the direction of the momentum density  $\rho\vec{v}$ , and the direction of the flux, which is just the velocity  $\vec{v}$ . As a matter of principle, you can always spot the directions of any tensor defined through a sequence of outer products of vectors by looking at the direction of each vector. In fact, both of those directions are the same in this specific case, since  $\rho\vec{v} \otimes \vec{v}$  and  $\vec{v}$  point in the same direction!

So now that we know what that tensor means, we can think a bit more concretely about how to represent it mathematically. Well, just like the velocity can be represented using three components with reference to some arbitrary coordinate system, the momentum density flux is the product of every component of the 3-D vector  $\rho\vec{v}$  with every component of the 3-D vector  $\vec{v}$ , so that the tensor  $\rho\vec{v} \otimes \vec{v}$  is represented by  $3 \times 3 = 9$  independent components. To make sure we distinguish which direction is which, mathematicians usually visually represent second-order tensors using a visual "square" of numbers called a matrix.

This isn't the end of the story on tensors, though; remember that the Cauchy momentum equation we saw before includes the term  $\nabla \cdot (\rho\vec{v} \otimes \vec{v})$ , not just the momentum density flux. Hence, we need to be able to understand the effects of trying to take derivatives, like the divergence and gradient, on tensors. To do that, let's repeat our formula on testing it out on the velocity—which we know is a

## Momentum density flux

Flux →

← Momentum density	$\rho v_x v_x$	$\rho v_x v_y$	$\rho v_x v_z$
	$\rho v_y v_x$	$\rho v_y v_y$	$\rho v_y v_z$
	$\rho v_z v_x$	$\rho v_z v_y$	$\rho v_z v_z$

Figure 07.2: These are the nine different components of the momentum density flux tensor, associating every component of the momentum density with every component of the flux. They've been placed in a geometric array called a matrix that highlights the different directions associated with each part of the momentum density flux tensor.

first-order tensor—and see how to extrapolate from there.

The divergence of the velocity at a point, from a purely mathematical standpoint, tells you what the rate of change of the velocity is at that point as you move in each of those predefined spatial coordinates, and then adds all of those rates of changes up into a single direction-less number that tells you if the flow is "accumulating" or "diminishing" at that point. Make sure to notice that the divergence of the velocity is truly directionless; if you looked at a flow upside down,

the direction of the flow would certainly change, but the amount by which the flow is "accumulating" at a point doesn't. This is easily extrapolated to a general, tensorial case; the divergence of a tensor takes the spatial rate of change of the tensor in the direction of one of the tensor's directional elements, usually defined to be the "last" element, and adds them up. This causes a tensor to "lose" a direction when acted upon by the divergence—meaning that the second-order tensor  $\rho \vec{v} \otimes \vec{v}$ , when acted on by the divergence, gives you a first-order tensor/vector, just as the momentum transport equation required.

Adding up this way  $\rightarrow$

Flux  $\rightarrow$

$\leftarrow$  Momentum density

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

$\nabla \cdot (\rho \vec{v} \otimes \vec{v})$

Figure 07.3: The divergence of the momentum density flux tensor (order 2). Here, we take the spatial rate of change of each of the 9 elements in the flux direction, and add them up to obtain 3 elements representing a vector (order 1). Note the directions of the partial derivatives match the directions of the flux velocities.

Lastly, let's take a look at the gradient of the velocity  $\nabla \vec{v}$ . Here

we are still obtaining the spatial rates of change of the velocity, but this time we're not adding anything up; in fact, we're obtaining the spatial rate of change of every component of the velocity in each and every one of those arbitrarily defined directions of space. 3 different possible rates of change for each of the three components indicates that the velocity gradient possesses 9 distinct pieces of information, revealing that the gradient of the velocity is a second-order tensor. As such, we can extrapolate to the general case and state that the gradient of a tensor is another tensor of a single higher order.

**Velocity gradient**

Gradient →

← Velocity	$\frac{\partial v_x}{\partial x}$	$\frac{\partial v_x}{\partial y}$	$\frac{\partial v_x}{\partial z}$
	$\frac{\partial v_y}{\partial x}$	$\frac{\partial v_y}{\partial y}$	$\frac{\partial v_y}{\partial z}$
	$\frac{\partial v_z}{\partial x}$	$\frac{\partial v_z}{\partial y}$	$\frac{\partial v_z}{\partial z}$

Figure 07.4: The velocity gradient tensor. The two "directions" associated with this tensor are connected to the velocity and to the direction of the rate of change.

Explicitly defining the direction that gets added as a result of

taking the gradient is trickier to spot than in the outer product case, where the directions of the resulting tensor are just the directions of each component of the tensor; but the direction exists, and the vector representing it can be found by solving the following equation for the vector  $\vec{m}$ :

$$\nabla \vec{v} \cdot \vec{m} = \vec{v}$$

(The dot product here is just representing standard matrix-column vector multiplication.) This vector doesn't really have a name, nor a more intuitive description other than the above as far as I can tell; but I call it the gradial vector for book-keeping, and would love to find a nice application for it.

These explanations are by no means rigorous or comprehensive; there are many incredible and beautiful aspects to the study of tensors that I sadly have to exclude for the sake of coherence, but I invite you to read up on them if you're interested in the subject. My current favorite book on the topic is Henry Block's *Introduction to Tensor Analysis*, which is tragically out-of-print but has been preserved electronically by the thoughtfulness of faculty members in the Theoretical & Applied Mechanics field at Cornell. In any case, this is just about everything we need to know about tensors to get a handle on all the mathematical machinery abound in introductory fluid mechanics; and although the principal language of fluid mechanics is vector calculus, it's good to get a handle on tensors to deal with an often-

ignored part of the world of fluid mechanics—the process by which molecules exert forces on each other when a fluid moves, known as the study of rheology.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. What is the difference between a second-order tensor and a matrix? Are all matrices second-order tensors? Are all second-order tensors matrices?
3. If a fluid is incompressible, how does the expression for  $\nabla \cdot (\rho \vec{v} \otimes \vec{v})$  simplify? Remember the incompressibility equation,  $\nabla \cdot \vec{v} = 0$ .
4. How would you describe the gradient of a second-order tensor? Extrapolate from the principles shown here for the velocity gradient.
5. Consider the gradient of any simple scalar (order 0 tensor) function. If you changed the coordinate axes, how would the gradient change? Would it change in a different way than a velocity vector would?

6. What happens if you take the divergence of the gradient of the velocity? How is that different from taking the gradient of the divergence of the velocity? The first expression is called the vector Laplacian, and will feature prominently in the following sections.

## Further Reading & Context

More precise and formal introductions to the idea of a tensor than presented here in the contexts of fluid mechanics can be found in [3] and [34]. Introductions in the context of general continuum mechanics can be found in [64] and [45]. A fully mathematical introduction to the construction of tensors independent of physical context can be found in [56]. My personal favorite introductory reference, which balances generality with approachability, is [8], closely followed by [9]. My representation of the velocity gradient is chosen so that it matches traditional mathematical definitions of the gradient operation and the standard index formalism in linear algebra—many texts use a representation that is the transpose of this one ([33]

08

## RHEOLOGY



*"The Persistence of Memory", Salvador Dalí.*



As we saw during our discussion on transport theory, the principal equation of fluid dynamics really boils down to a transport equation for the momentum in a fluid, or for the transport of momentum density through fluid points:

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) = \vec{f} + \vec{r}_{mol}$$

where the  $\vec{r}_{mol}$  term represents forces stemming from molecular effects. It is this molecular term which exclusively "identifies" a fluid, as everything else in the governing equation above was generated through a very general mathematical framework describing the movement of "things" in space rather than through specific physical insight associated with fluids. Consequently, understanding this connection between continuum-scale momentum and molecular effects is a humongously important task for physicists and engineers alike, and is the principal endeavor of rheology. It's often extremely difficult to generate refined models for  $\vec{r}_{mol}$  from first principles, so often the process of determining an  $\vec{r}_{mol}$  for a given fluid consists of making some basic assumptions about the fluid and then testing the fluid experimentally to see if the behavior of the fluid is consistent with those assumptions. Rheology, as a result, is largely an experimental science. This doesn't mean we're simply left to fiddle around with rheometers from now on, however—in fact, we'll rapidly find that a couple of simple assumptions and some not-so-simple tensor

calculus will lead us to an equation that applies to nearly all fluids.

Firstly—and as we saw in Chapter 1—when we talk about the density or velocity of a fluid at a point, we're really talking about the average velocity or mass/volume ratio inside a small "molecular net" around a point. One thing we haven't talked about yet is on the consequences of formulating a theory of fluid mechanics strictly on these average quantities, and how fluctuations around those averages affects the way fluid molecules exert forces on each other at the macroscale. In order to do so, we first need to make a relatively sensible hypothesis about the nature of these molecular forces, a hypothesis I call the "fluctuation hypothesis"; that all random fluctuations of fluid quantities occur as a result of intermolecular forces in the fluid. Based on this, we can try to formulate the Cauchy momentum equation we derived before that excludes an implicit intermolecular force term—but for a randomly fluctuating density and velocity—and equate any differences we see between either momentum equation to intermolecular forces.

Let's start by defining a "true" fluctuating density  $\rho^*$  and velocity  $\vec{v}^*$  at a point, both of which fluctuate independently around an average  $\rho$  and  $\vec{v}$ . This lets us split up the true quantities into an average part, which we've been dealing with from the beginning, and a fluctuating part (indicated with  $'$ ) that is on average zero:

$$\rho^* = \rho + \rho'$$

$$\vec{v}^* = \vec{v} + \vec{v}'$$

Since these quantities are transported in a fluid just like their averaged counterparts, we can construct a momentum equation based on them that is just as valid as the one we saw before:

$$\frac{\partial (\rho^* \vec{v}^*)}{\partial t} + \nabla \cdot (\rho^* \vec{v}^* \otimes \vec{v}^*) = \vec{f}$$

Expanding out in terms of average and fluctuating components leads to a lengthy, messy expression:

$$\begin{aligned} \frac{\partial (\rho \vec{v})}{\partial t} + \frac{\partial (\rho' \vec{v}')}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) + \nabla \cdot (\rho' \vec{v} \otimes \vec{v}) + \nabla \cdot (\rho \vec{v}' \otimes \vec{v}) \\ + \nabla \cdot (\rho \vec{v} \otimes \vec{v}') + \nabla \cdot (\rho' \vec{v}' \otimes \vec{v}) + \nabla \cdot (\rho' \vec{v} \otimes \vec{v}') + \nabla \cdot (\rho \vec{v}' \otimes \vec{v}') \\ + \nabla \cdot (\rho' \vec{v}' \otimes \vec{v}') = \vec{f} \end{aligned}$$

This just looks like a big jumble of symbols, so we have to get to work on simplifying this a little bit. Luckily, we can get rid of nearly everything in one fell swoop by averaging everything out! This will get rid of every term that contains a single fluctuating term (since those terms are normal numbers multiplied by something that's on average zero) and everything that has a  $\rho' \vec{v}'$  in it (since both are independent and average to zero, they multiply to something that's on average zero). This results in a far simpler expression:

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) + \nabla \cdot (\overline{\rho \vec{v}' \otimes \vec{v}'}) = \vec{f}$$

where the  $\overline{\phantom{x}}$  indicates an average of something that wasn't already defined as an average. Curiously, the  $\overline{\rho \vec{v}' \otimes \vec{v}'}$  doesn't zero out! This

is because even though each individual  $\vec{v}'$  is on average zero, the combination  $\vec{v}' \otimes \vec{v}'$  isn't, since it's "kind of" the square of an individual  $\vec{v}'$ . The presence of that extra term shows that considering fluctuations in fluid mechanical quantities actually does lead to a verifiable change in the momentum transport equations we've been setting up! And since we had attributed every possible extra term in the fluctuating form of the Cauchy momentum equation to intermolecular forces—that was our fluctuation hypothesis—then for both momentum equations to match, it has to be true that:

$$\vec{r}_{mol} = -\nabla \cdot (\overline{\rho \vec{v}' \otimes \vec{v}'})$$

This gives us a look at how molecular-scale phenomena in a fluid leads to forces at the macroscale, but doesn't go very far in actually giving us a relationship between those molecular quantities and the averaged-out properties of a fluid we use to formulate fluid mechanics. Remember,  $\vec{v}'$  could be basically anything! So what fluid mechanicians do is collect everything inside of the parenthesis, flip the sign, and label the grouped object as a new, unknown second-order tensor  $\sigma$  commonly referred to as the stress. This leads to the form of the Cauchy momentum equation that most engineers & scientists use:

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) = \vec{f} + \nabla \cdot \sigma$$

Equivalently, the stress tensor  $\sigma$  is just a second-order tensor

that represents, for a given fluid point, the traction/force per unit area  $\vec{t}$  acting on that element thanks to its neighboring fluid point in the  $\hat{n}$  direction. If you've studied solid mechanics, you've likely heard the term stress thrown around before; it means the same thing there as it does here. In fact, the only difference between a solid and a fluid is that the stress is a function of different continuum parameters in each type of substance. Notice we technically never did anything fluid-specific when we derived the momentum transport equation in Chapter 5! Luckily, we already know one piece of the puzzle of  $\sigma$ ; it has to contain the forces coming from pressure gradients, since we spent two chapters discussing how pressure (and the forces it induces) is an entirely molecular effect. We can make our lives easier and pull that pressure gradient term out, making sure we make it negative so that everything is consistent with the hydrostatic equation we derived in Chapter 2:

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) = \vec{f} - \nabla p + \nabla \cdot \boldsymbol{\tau}$$

That new tensor  $\boldsymbol{\tau}$  is usually called the deviatoric stress tensor, which I think is a silly name. The contents of the deviatoric stress tensor are in some sense what we're really looking for, since the notion of pressure and pressure gradients is universal in solids and liquids. And here, we can classify fluids rheologically into two important & distinct branches based on a key characteristic of their behavior

when standing still; fluids whose static behavior is fully characterized by pressure gradients (commonly called simple fluids), and fluids for which it is not (commonly called non-Newtonian fluids). For the latter, the molecules in the fluid will usually have some kind of extra molecular force that "causes" them to try to remain in some specific macroscopic shape or configuration—just like a solid. Some examples of this are ketchup (it doesn't like to flow out unless you smack the bottle or pump it out), mucus, pancake mix, melted rubber, etcetra; there's too many to count. A simple experimental way to figure out whether or not a fluid is simple or non-Newtonian is to pour it into onto a surface and then tilt the surface very, very slightly; if the fluid flows at even a slight angle, then it's safe to say the fluid behaves like a simple fluid. (Professionals use rheometers). Modeling non-Newtonian fluids is very tricky, and usually done on a case-by-case basis; we won't concern ourselves here with understanding how they're modeled, other than that their additional static molecular forces usually depend on the same thing that they depend on for solids, which is some displacement off of an "equilibrium" configuration like in a spring. Looking back at simple fluids, modeling them is far easier than modeling non-Newtonian fluids since the assumption on the behavior of simple fluids induces quite a big constraint on that deviatoric stress tensor  $\tau$ ; if we take the assumption that every molecular effect that induces forces in an unmoving fluid is by

definition captured through pressure gradients, then we find that  $\tau$  can't possibly contain any terms involving the physics of the fluid when it's standing still. It is entirely a function of the properties of the fluid's motion, which is why I like to call  $\tau$  the hydrodynamic stress tensor when modeling simple fluids.



Figure 08.1: The "tilt test"; a puddle of simple fluid in blue and a puddle of non-Newtonian fluid in yellow are placed on a surface. When the surface is tilted slightly, the simple fluid loses its form and flows while the non-Newtonian fluid deforms but doesn't flow/preserves form.

Looking back at simple fluids, modeling them is far easier than modeling non-Newtonian fluids since the assumption on the behavior of simple fluids induces quite a big constraint on that deviatoric stress tensor  $\tau$ ; if we take the assumption that every molecular effect that induces forces in an unmoving fluid is by definition captured through pressure gradients, then we find that  $\tau$  can't possibly contain any terms involving the physics of the fluid when it's standing still. It is entirely a function of the properties of the fluid's motion, which is why I like to call  $\tau$  the hydrodynamic stress tensor when modeling simple fluids.

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) = \vec{f} - \nabla p + \nabla \cdot (\boldsymbol{\tau}[\nabla \vec{v}])$$

Here the brackets just indicate that  $\boldsymbol{\tau}$  is a function of  $\nabla \vec{v}$ . But remember; when talking about velocity gradients, we have to consider the gradients in each direction of the velocity components in each direction. So instead of representing three pieces of independent information like  $\vec{v}$  does, velocity gradients represents nine, meaning that it's—you guessed it—a tensor. As a matter of fact, the assumption above puts a further restriction on  $\boldsymbol{\tau}$ ; if I decided to somehow spin myself around a static fluid I'm observing in the lab, I would observe a nonzero velocity gradient even though the fluid is standing perfectly still. As a result, a simple fluid's deviatoric stress tensor can't depend on every part of the velocity gradient tensor, just some parts. Some behind-the-scenes tensor manipulations show that in a simple fluid, the deviatoric stress tensor can only depend on the symmetric part of the velocity gradient tensor. This property is commonly referred to as objectivity, and the symmetric part of the velocity gradient tensor is usually called the strain-rate tensor  $\mathbf{E}$ , calculated from the velocity gradients by:

$$\mathbf{E} = \frac{\nabla \vec{v} + \nabla \vec{v}^T}{2}$$

This still doesn't really get us anywhere; the nine components of  $\boldsymbol{\tau}$  are so far completely arbitrary functions of the nine components of  $\mathbf{E}$ .



But we can section simple fluids out into two other categories; fluids for which  $\tau$  is strictly proportional to the strain-rate tensor, which are commonly called Newtonian fluids, and fluids for which it is not, which are called generalized Newtonian fluids. (For what it's worth, I think the naming conventions for Newtonian, non-Newtonian, and generalized Newtonian fluids is extremely confusing, but there's not much I can do about it.) Examples of generalized Newtonian fluids include blood, nail polish, paint, syrup, etcetra. For generalized Newtonian fluids, the behavior of the fluid is properly "liquid", but the magnitude of the molecular forces induced by the flow of the fluid are complicated functions of the components of the gradients in the velocity. Models for these functions are more of an art than a science, based largely on experimental results. For a Newtonian liquid, however, we now find that there is a linear relationship between the second-order strain-rate tensor  $\mathbf{E}$ , with nine independent components, and the second-order deviatoric stress tensor  $\tau$ , also with nine independent components. This means that we can represent the linear relationship by a fourth-order tensor with 81 different components (yikes!) which I'll call the viscosity tensor  $\mathbf{M}$ :

$$\tau = \mathbf{M} : \mathbf{E}$$

For better or worse, there's an extra assumption that nearly always applies to fluids of this type; the molecular interactions are indepen-

dent of direction, and as a result the stress tensor is independent of the orientation of the velocity gradients. Fluids for which this holds true are called isotropic, and fluids for which it's not are called anisotropic. I've never heard of an anisotropic simple Newtonian fluid, but maybe you'll be the first to find one! For a simple Newtonian isotropic fluid, doing some behind-the-scenes tensor calculus reveals that our updated conservation of momentum equation has to take the following form once we all these assumptions:

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) = \vec{f} - \nabla p + \mu \nabla^2 \vec{v} + (\lambda + \mu) \nabla (\nabla \cdot \vec{v})$$

This absolute mess of an equation is called the compressible Navier-Stokes equation, and is the equation nearly everyone uses when dealing with flow of compressible fluids. The two symbols  $\mu$  and  $\lambda$  are undetermined proportionality factors that are the only surviving components of that viscosity tensor  $\mathbf{M}$ , and are usually determined through experiment; the first is universally referred to as the dynamic viscosity or just viscosity, and the second is so inconsistently defined in the literature that the only name we can give this thing that rheologists wouldn't argue over is the profoundly unhelpful moniker of second Lamé parameter. If we further assume that the fluid is incompressible and that the viscosity is the same everywhere, we can polish the equation up a lot more and pull the density out of a lot of expressions, get rid of that last term that depends on  $\lambda$ , and wind up

with:

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) = \vec{f} - \nabla p + \mu \nabla^2 \vec{v}$$

This is the incompressible Navier-Stokes equation, also known as just the Navier-Stokes equation, and it is the most famous equation in fluid dynamics; understanding it and analyzing its components will be the sole subject of the next chapter. Given how involved this sequence of assumptions is for an introduction to fluid mechanics, I've put a table below describing how each assumption we made brought us from the Cauchy momentum equation at the beginning of this chapter to the Navier-Stokes equation above.

Assumption	Mathematical Consequence	When It's Wrong
Fluctuation hyp.	$\vec{r}_{mol} = -\nabla \cdot (\rho \vec{v}' \otimes \vec{v}')$	Exotic molecular interactions
Simplicity	$\vec{r}_{mol} = -\nabla p + \nabla \cdot \boldsymbol{\tau}$	Material has solid-like properties, or pressure is defined differently
Objectivity	$\boldsymbol{\tau} = \boldsymbol{\tau}[\mathbf{E}]$	Velocities/rotations are extremely fast
Linearity	$\boldsymbol{\tau} = \mathbf{M} : \mathbf{E}$	Molecular forces have a complicated relationship with velocity gradients
Isotropy	$\boldsymbol{\tau} = 2\mu \mathbf{E} + \lambda \text{Tr}(\mathbf{E}) \mathbf{I}$	I don't know (molecules would likely have to be very asymmetric)
Incompressibility	$\boldsymbol{\tau} = \mu \nabla^2 \vec{v}$	Density is very low, fluid speeds are comparable to molecular speed (see Chapter 2)

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. What physical effects do you think the viscosity and second Lamé parameter quantify? Can you describe flows where only one of the terms is nonzero?
3. Can you think of other ways to test if a fluid is simple or not? How about to test if a fluid is Newtonian or not?
4. If you're brave enough to handle some tensors, can you find why the form of the deviatoric stress tensor goes from  $2\mu\mathbf{E} + \lambda\text{Tr}(\mathbf{E})\mathbf{I}$  to  $\mu\nabla\vec{v}$  when we take the incompressibility assumption? Remember the definition of the strain-rate tensor!
5. What statements can you make about the stress tensor based just on the fact that  $\boldsymbol{\sigma} = -\rho\overline{\vec{v}' \otimes \vec{v}'}$ ?
6. What would the Navier-Stokes equation look like if the fluid was a simple Newtonian isotropic fluid but with a non-uniform viscosity?
7. Let's say that you could describe the forces per unit area in the definition of the stress as coming from an energy density

gradient. How would you define the stress in that case, and what would be the associated directions?

## Further Reading & Context

Some textbooks about fluid mechanics don't discuss rheology beyond the Navier-Stokes assumptions, but some—particularly those based on continuum mechanics in general, do ([29][64][3]). A general introduction to rheological concepts can be found in [4] and in [39]. A short introduction to rheological modeling via constitutive equations can be found in [3]. A comprehensive description of experimental rheology can be found in [13], while a more modern description (albeit at the microscale) can be found in [20]. An illuminating discussion of the meaning of the Lamé parameters in relation to definitions of pressure, both historical and mathematical, can be found in [67].

Although the presentation I employ in this text to derive the existence of a stress tensor is non-standard and a bit mathematically sloppy—the "fluctuation hypothesis" nomenclature is introduced in this text & certain continuity assumptions need to be made on the fluctuating quantities for this all to be rigorous, for example—its results are fully equivalent to the rigorous Chapman-Enskog construction employed in [46] and [27], and produces no conflict to the standard "Cauchy tetrahedron" formulation found in continuum me-

chanics or mathematically advanced fluids texts ([64][45][47][41][37]). In the "Cauchy tetrahedron" formulation, the mathematical plausibility of formulating surface tractions as the divergence of a stress tensor is discussed, and this stress tensor is then connected to molecular effects via constitutive hypotheses. See [41] for a short discussion on the effects of molecular interaction failing to be local in this context.

09

# NAVIER-STOKES, EXISTENCE & UNIQUENESS



*"Dynamism of a Car"*, Luigi Russolo.

At long last, we've finally managed to derive the famous Navier-Stokes equation; the cornerstone of most of fluid mechanics. Notice all the assumptions we had to make to get to it! To refresh your memory, this is what it looks like, along with descriptions of each term in it:

$$\begin{array}{ccccccc}
 \begin{array}{c} \text{Rate of change} \\ \text{of momentum} \\ \text{density with} \\ \text{time} \end{array} & & \begin{array}{c} \text{Net flow of} \\ \text{momentum density} \\ \text{in/out of a point} \end{array} & & \begin{array}{c} \text{Intermolecular} \\ \text{pressure-induced} \\ \text{force density} \end{array} & & \begin{array}{c} \text{External force} \\ \text{density (gravity,} \\ \text{electric forces,} \\ \text{etc.} \end{array} \\
 \frac{\partial(\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v}) = -\nabla p + \underbrace{\nabla^2 \vec{v}}_{\substack{\text{Intermolecular} \\ \text{flow-induced} \\ \text{force density}}} + \vec{f}
 \end{array}$$

Figure 09.1: The Navier-Stokes equation, which describes momentum density transport in a simple, Newtonian, isotropic, incompressible fluid. The terms on the left-hand side are mathematical consequences of the concept of transport, while the terms on the right-hand side (save for the external force density) are defined by intermolecular fluid interactions.

There are many different, equivalent ways to write the Navier-Stokes equation, but the most common is this one, which incorporates a couple of simplifications thanks to conservation of mass:

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

(Note that some people write the dot product in the flux term back-



wards based on their notation for the gradient, but it's all just preference.) Another equivalent form I briefly mentioned when discussing transport theory, which is particularly useful when discussing the curvature of flows, is commonly known as the Lamb form:

$$\rho \left( \frac{\partial \vec{v}}{\partial t} + \frac{\nabla (\vec{v} \cdot \vec{v})}{2} - \vec{v} \times (\nabla \times \vec{v}) \right) = \vec{f} - \nabla p - \mu \nabla \times (\nabla \times \vec{v})$$

In whatever form it takes, this equation (along conservation of mass and varying simplifying assumptions) will be the chief mathematical tool we'll use to determine the shape and speeds of flows. That being said, it turns out that the Navier-Stokes equation is notoriously difficult to deal with except in the simplest of situations, and it's all thanks to the flux term:

$$\rho \nabla \vec{v} \cdot \vec{v}$$

This little mathematical pest has been the bane of fluid mechanicians for a solid hundred years, and it induces such a monumental headache on any mathematician attempting to solve the Navier-Stokes equation that there is actually a million-dollar bounty out for anyone who is even able to prove that the Navier-Stokes equation always has "sensible" solutions for a physical input—this is one of the most important mathematical problems ever conceived! The reason this flux term is such a problem is because of a property called non-linearity, which is best shown rather than told. To demonstrate, let's consider a silly non-physical problem in 1-D, where we consider two

imaginary fluids whose velocity is solely determined by the following equations:

	Fluid 1	Fluid 2
1-D Equation	$\rho v \frac{dv}{dx} = f$	$\mu \frac{d^2 v}{dx^2} = f$
3-D Equivalent	$\rho \nabla \vec{v} \cdot \vec{v} = \vec{f}$	$\mu \nabla^2 \vec{v} = \vec{f}$

Applying some calculus tricks, what you'll wind up finding is that the expressions for the velocities in each of the fluids is markedly different:

	Fluid 1	Fluid 2
1-D Solution	$v(x) = \pm \frac{\sqrt{2} \sqrt{c_1 \rho + f x}}{\sqrt{\rho}}$	$v(x) = c_2 x + c_1 + \frac{f x^2}{2\mu}$

Note that for Fluid 2, if I set the constants to 0, the velocity is linearly proportional to the force. This means that if I double the force, the velocity everywhere is going to be doubled, and so on—a property unsurprisingly called linearity. For Fluid 1, however, this isn't true; setting the constant to 0 and doubling the force instead leads to an increase in the velocity by a factor of  $\sqrt{2}$ ! As innocuous as it seems, nearly every single tool ever made by mathematicians and scientists to solve partial differential equations like the Navier-Stokes equation relies on linearity, meaning any attempts to solve the Navier-Stokes equation relies on conceptual machinery on the outside of most of mathematical history. But that's not even the

worst part; as it turns out, nonlinearity is often comorbid with a host of horrendous properties, and the velocity of Fluid 1 is no exception.

For example, consider what would happen if I attempt to find the flow velocity for Fluid 1 if I take the velocity at the origin to be zero,  $v(0) = 0$ ; because of that  $\pm$  sign in front of the velocity expression for Fluid 1, I actually have two possible fluid velocities! This is incredibly strange from a physical perspective; one could conjecture a situation where I have two exact copies of this system, where I'm applying the same force under the same conditions to each of them, and somehow get different flows. This goes straight against any rational understanding of physics, and is called non-uniqueness. Even worse, no matter what value of the constant  $c_1$  I choose based on some physical information, or which expression for the velocity of Fluid 1 I pick, there is always some value of  $x$  past which my velocity suddenly turns into an imaginary number! This means that in some regions, we don't even have an expression for the flow velocity, which is even more mind-boggling from a physical perspective and is referred to by mathematicians as existence failure (this is a really cool name). And this is only in one dimensions, folks—just imagine how much worse it gets in 3.

When you include time into the equations, things get even worse; you can start off with a flow that makes sense, and find it evolving into something seemingly nonsensical! Take, for example, what happens

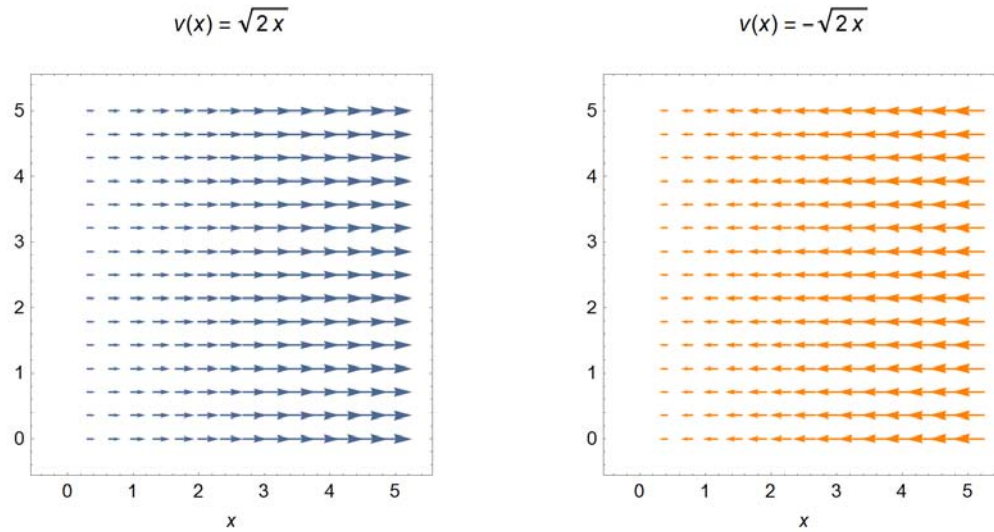


Figure 09.2: Two solutions for the velocity of Fluid 1, each solving  $v(x)\frac{dv(x)}{dx} = 1$  with zero velocity at the origin, corresponding to a fluid with  $f, \rho = 1$ . Neither solution is defined in the region  $x < 0$ .

if you now take the ODE describing Fluid 1 and add a time rate-of-change term, turning it into a PDE:  $\frac{\partial v}{\partial t} + \rho v \frac{\partial v}{\partial x} = f$  This equation represents momentum transport in a one-dimensional fluid with no intermolecular interactions, and is commonly called Burgers' equation. Take a look at what happens if you take a perfectly nice initial flow field for such a fluid and let it evolve in time for a constant unit  $f$  and  $\rho$ :

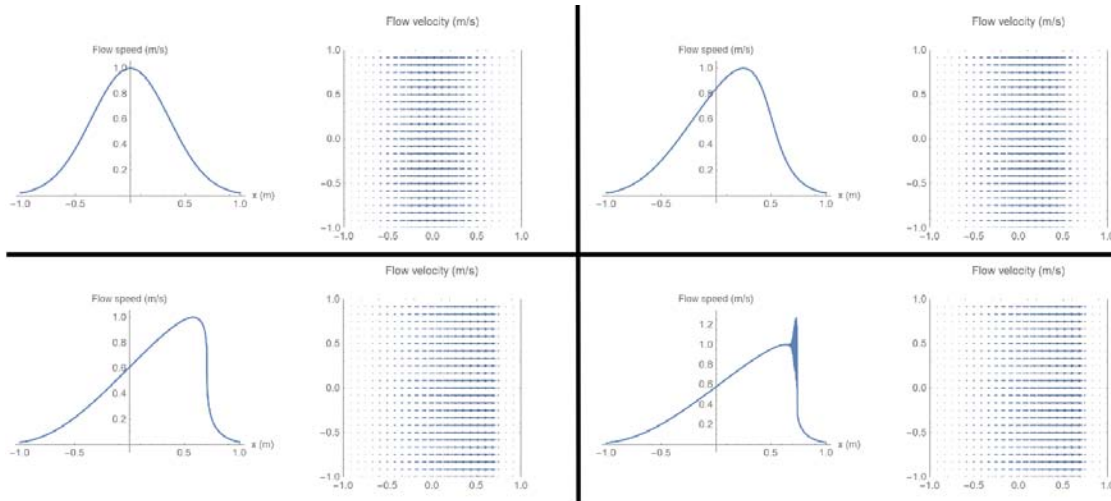


Figure 09.3: The solution to the PDE  $\frac{\partial v}{\partial t} + v \frac{\partial v}{\partial x} = 1$  for a smooth initial condition. The function describing the velocity eventually develops a discontinuity in space, causing the solution to become ill-defined. The numerical solver fails to handle this phenomenon, and glitches out once it develops.

After time passes, the velocity becomes discontinuous, and jumps immediately from one value to another; a phenomenon commonly referred to as a shock or shock wave. Mathematically, this is a problem not just because  $\frac{dv}{dx} = \infty$  at the discontinuity, but because the velocity has effectively two values there, causing it to be ill-defined! What happens here, from a philosophical perspective, is astounding; we've gone and defined an object mathematically using the rules of calculus, and the object has then evolved over time into something that violates the very mathematical principles we used to define it. This is exactly the problem (or at least, one of them) people run into when trying to solve the Navier-Stokes equation; the objects one uses

to define flows for fluids can very rapidly outgrow the conceptual framework we used to define them.

Consequently, nearly every analytical attempt at using the Navier-Stokes equations to describe a flow necessitates ensuring that the momentum flux term in the equations is either negligible or handled indirectly. One way to verify this negligibility is by taking the ratio of the magnitudes of each component of the flux term with the magnitudes of the components of the only other term that involves spatial gradients of the velocity—the viscosity term:

$$\mathbf{Re} = \rho \nabla \vec{v} \cdot \vec{v} \oslash \mu \nabla^2 \vec{v}$$

The  $\oslash$  symbol represents that we are dividing every term of a tensor by every other term of another tensor, effectively acting like the division counterpart of the outer product  $\otimes$ . Since each of these quantities is a 3-D vector, this ratio actually contains 9 distinct ratios, and I refer to it as the Reynolds tensor. The Reynolds tensor is useful in the sense that it is telling us whether or not, for a given flow, we should expect to observe the same kind of mind-boggling behavior we saw above in the Burgers' equation, where it would be located, and in what directions we expect it to manifest. Because calculations involving the Reynolds tensor are pretty unwieldy, fluid mechanicians almost always single out a specific component of the Reynolds tensor, guess "characteristic" values for the density/viscosity/velocity/velocity gra-

dients/lengths that they assume are valid everywhere in the flow, and come up with a single scalar ratio called the Reynolds number which they use instead. Sometimes this leads to spurious assumptions and misdrawn conclusions, but alas.

## Re

Flow-induced intermolecular forces →

← Net momentum density flux	$\frac{\rho \left( v_x \frac{\partial v_x}{\partial x} + v_y \frac{\partial v_x}{\partial y} + v_z \frac{\partial v_x}{\partial z} \right)}{\mu \left( \frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2} \right)}$	$\frac{\rho \left( v_x \frac{\partial v_x}{\partial x} + v_y \frac{\partial v_x}{\partial y} + v_z \frac{\partial v_x}{\partial z} \right)}{\mu \left( \frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2} \right)}$	$\frac{\rho \left( v_x \frac{\partial v_x}{\partial x} + v_y \frac{\partial v_x}{\partial y} + v_z \frac{\partial v_x}{\partial z} \right)}{\mu \left( \frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2} \right)}$
	$\frac{\rho \left( v_x \frac{\partial v_y}{\partial x} + v_y \frac{\partial v_y}{\partial y} + v_z \frac{\partial v_y}{\partial z} \right)}{\mu \left( \frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2} \right)}$	$\frac{\rho \left( v_x \frac{\partial v_y}{\partial x} + v_y \frac{\partial v_y}{\partial y} + v_z \frac{\partial v_y}{\partial z} \right)}{\mu \left( \frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2} \right)}$	$\frac{\rho \left( v_x \frac{\partial v_y}{\partial x} + v_y \frac{\partial v_y}{\partial y} + v_z \frac{\partial v_y}{\partial z} \right)}{\mu \left( \frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2} \right)}$
	$\frac{\rho \left( v_x \frac{\partial v_z}{\partial x} + v_y \frac{\partial v_z}{\partial y} + v_z \frac{\partial v_z}{\partial z} \right)}{\mu \left( \frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2} \right)}$	$\frac{\rho \left( v_x \frac{\partial v_z}{\partial x} + v_y \frac{\partial v_z}{\partial y} + v_z \frac{\partial v_z}{\partial z} \right)}{\mu \left( \frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2} \right)}$	$\frac{\rho \left( v_x \frac{\partial v_z}{\partial x} + v_y \frac{\partial v_z}{\partial y} + v_z \frac{\partial v_z}{\partial z} \right)}{\mu \left( \frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2} \right)}$

$$Re = \frac{\rho V L}{\mu}$$

Figure 09.4: The components of the Reynolds tensor in Cartesian coordinates, along with a typical Reynolds number-type approximation. Calculating the Reynolds tensor is often unwieldy, so most scientists simply estimate a single number based on quantities in the problem they expect to see and call it the Reynolds number, shown below.

Nearly all of the time, we use the Reynolds tensor/number to do sanity checks on any simplifying assumptions we make to the

Navier-Stokes equation—either by using a conjectured Reynolds tensor/number to justify removing some term from the Navier-Stokes equations, or by calculating the Reynolds tensor/number for a solution to a simplified Navier-Stokes equation to validate whether or not the simplification was sensible in the first place. And one setting where we'll be able to get rid of that flux term and, as a result, easily construct specific conclusions about flow shapes and speeds is a setting we've looked at before—pipe flow.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. Some people think adding a term representing viscosity to equations like the ones described above help get rid of the shocks & other strange behaviors we saw just now. Try solving the ODE  $\rho v(x) \frac{dv}{dx} - \frac{d^2v}{dx^2} = f$  for a bunch of different  $\rho/f/\mu$  using some math computer programs to see whether or not that line of thinking is correct.
3. How would neglecting the flux term in a problem change the physics of the fluid you would observe, and would it make sense? Is there a fluid that can possess a zero flux term?



4. In what situations do you think simply using the Reynolds number instead of the Reynolds tensor would be sufficient? In what situations would it fail?
5. Try calculating the Reynolds tensor for simple flows; maybe a constant flow, or a flow that changes linearly in one direction in space. What components are zero, and which aren't? What do they correspond to?
6. Shock waves appear to make no sense in a differential equations sense, but seem to make perfect sense physically; is there some mathematical tool we were using before that would be able to make mathematical sense of shock waves?
7. In Chapter II, we talked about how pressure ensures liquid doesn't just accumulate into a hyperdense thin film at the bottom of a cup. How is that connected to the shock wave phenomenon we saw above?
8. If you want to get a glimpse of how linearity is useful for solving differential equations, solve  $\frac{dv}{dt} = f$  for  $f = 1$  and  $2$  for arbitrary constants. Can the sum of both of these solutions represent a solution for  $\frac{dv}{dt} = 3$ ?

## Further Reading & Context

An approachable introduction to the idea of existence & uniqueness of solutions of differential equations can be found in [28]. The study of existence & uniqueness of solutions to the Navier-Stokes equation is an active and prominent field of study in mathematics as of 2020—it is key to note that the examples provided in this chapter are not for real fluids, as science has not created a complete or comprehensive picture of existence or uniqueness for Navier-Stokes fluids in 3-D. An excellent introduction to this subject can be found in [38]. An under-recognized pioneer of such studies was Olga Ladyzhenskaya, who proved the existence and uniqueness of solutions to the Navier-Stokes equations in 2-D in [35] and wrote extensively on the subject in works such as [36]. A fluid-dynamical system that obeys the Navier-Stokes equation which unambiguously possesses multiple solutions and is easily replicatable is flow between coaxial spinning cylinders—this is called Taylor-Couette flow, and is studied deeply in [10]. For an introduction to the behavior of shock waves in fluid mechanics, see [2]. For a mathematical introduction to the idea of modeling shocks in continuum systems obeying conservation laws, see [29].

The notion of a Reynolds tensor is non-standard, and is introduced in this text. A field of fluid dynamics in which predictions

based on a single Reynolds number can often be verifiably incorrect compared to experiment is in the flow of colloidal particles—see [63] or [57] for fluid-dynamical systems where the "obvious" Reynolds number fails to describe significant physical phenomena in the system.

# 10

## PIPE FLOW & TURBULENCE



*"Revolution of The Viaducts", Paul Klee.*

With the analytical tool of the Navier-Stokes equation in hand, we can finally begin to make statements about the precise shapes and speeds of flows. Perhaps the best place where we can make those statements without running afoul of the pathology of the Navier-Stokes equation—and trust me, we will run afoul of it—is in studying flow in pipes or channels, which we described using hydraulic equations in Chapter 5.

Let's again consider steady incompressible flow through a horizontal section of pipe with length  $L$  and with a uniform cross-section. This time, we're going to try and explicitly define what the momentum loss  $\vec{H}$  is using the Navier-Stokes equation, and get a formula for the shape and speed of the flow inside of the pipe as a function of the other free parameters in the system. To do this, we'll need to do a couple of tricks to bring everything down from the integral formulation we described in Chapter 5—the fundamental equations of hydraulics—to a differential one.

Now, the first fundamental equation of hydraulics tells us that, because the fluid is incompressible, conservation of mass requires that the speed at the inlet of the pipe be the same as the speed at the outlet, meaning that all the momentum coming into the pipe from flux at the inlet must leave at the outlet. In addition, all the momentum being added into the pipe due to gravity is acting perpendicular to the momentum loss; this immediately indicates that the

momentum loss in steady flow within such a pipe must necessarily be compensated by a drop in the pressure (and only a pressure drop) along the flow direction.

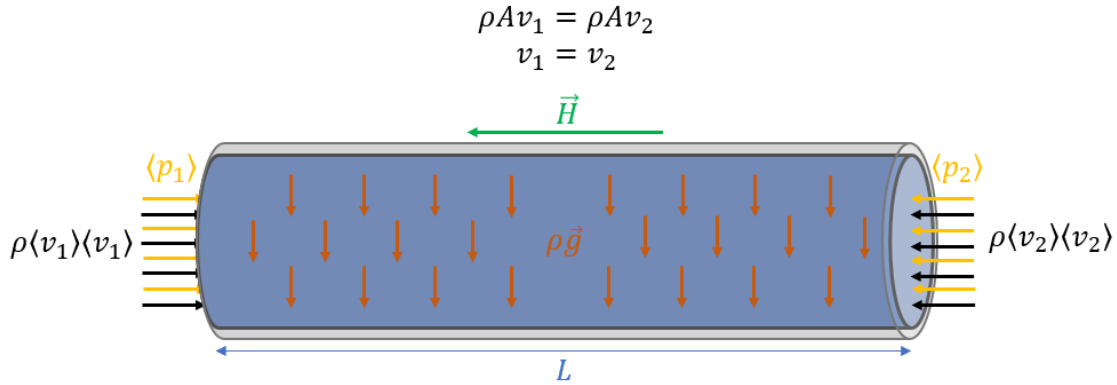


Figure 10.1: A horizontal pipe of length  $L$  with constant cross-sectional area and the associated tractions/force densities acting on it. Conservation of mass indicates that the speeds on either end are equal; this means the loss coming from  $\vec{H}$  can only be balanced by a drop in the pressure between the ends due to conservation of momentum.

A nice thing about this result is that it is totally independent of the length of the pipe section! We can make the pipe as long or as short as we want and the above statement still holds. Now I'm going to do a classic math trick; I'm going to pick some horizontal line across the pipe, and represent the pressure through that arbitrary line as a function of the distance from the inlet  $z$  with a Taylor series off of the value at the inlet portion of the line:

$$p = p_1 + G_1 z + G_2 z^2 \dots$$

If I wanted to represent the pressure drop, I could just subtract by the inlet value:

$$\Delta p = G_1 z + G_2 z^2 \dots$$

If I take the length of the pipe and shrink it down more and more, the squared term is going to get smaller than the linear term as long as both terms are nonzero. This means that, for an infinitesimally short length of pipe, the pressure drop is always strictly linear in the length, and so the pressure drop per length is a constant,  $G_1$ .

Now think about this; if I assume the momentum loss is independent of the pressure, I can take the section of pipe and split it up into a bunch of infinitesimally tiny segments one behind the other, which should all be essentially identical to each other; sure, the pressure at each of the inlets might be different, but the pressure drop should be the same, as well as everything else! This implies the pressure drop is linear in all of the tiny segments, each with the same pressure drop per length  $G_1$ , so the pressure drop per length in a finite section of such a pipe must be constant (under the given assumptions, of course).

This gives us what we need to analyze pipe flow through a horizontal section of pipe with constant cross-sectional area using Navier-Stokes. In this scenario, based on the argument above, we are going to impose some arbitrary pressure drop per length through the pipe

$G_1$ , and see what the resulting flow field looks like. The argument above guarantees (or at least, suggests) that imposing such a pressure drop is physically sensible, and will lead to a sensible solution of the Navier-Stokes equation. Before we do, however, notice a very curious quirk in our logic—if we assume the pressure drop per length is constant, that means that the pressure in the pipe through some arbitrary horizontal line is going to be of the form:

$$p = p_1 + G_1 z$$

where  $p_1$  can be the pressure in an arbitrary location of the pipe. No matter how big that arbitrary reference pressure is, there is always some length of pipe in one direction that causes the pressure to go negative, which makes no physical sense! In that sense, we find that any solution we get for such a problem can't apply to arbitrarily long pipes, as we will invariably run into a region of pipe where the solution we find simply cannot exist—an existence failure, as we saw in Chapter 8, and a harbinger of things to come.

Alright, let's get on with the solution already. We want to solve the incompressible Navier-Stokes equation for steady flow inside of a cylindrical pipe with a uniform cross-sectional pressure and a constant pressure drop per length in the "pipe" direction:

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



Let's list out a couple of observations and "common-sense" assumptions that get us to a point where we can handle this equation:

- We assumed the flow is steady already, so the time rate of change term is zero:  $\rho \frac{\partial \vec{v}}{\partial t} = 0$ .
- There's no flow in any direction other than in the "pipe direction" (a.k.a. the longitudinal direction); this means there's no flow in the radial direction or the angular direction (swirling flow):  
 $\vec{v} = [v_r \ v_\theta \ v_z] = [0 \ 0 \ v_z]$
- Everything about the flow is totally independent of the pipe angle, so we can kill off any terms involving changes in the angular direction:  $\frac{\partial \dots}{\partial \theta} = 0$
- Gravity is the only external force and acts downwards, so it only causes pressure gradients perpendicular to the flow direction.

If we take a look at the incompressible conservation of mass equation and take into consideration these assumptions, we see something that is fairly obvious:

$$\nabla \cdot \vec{v} = \frac{1}{r} \frac{\partial(rv_r)}{\partial r} + \frac{1}{r} \frac{\partial(v_\theta)}{\partial \theta} + \frac{\partial v_z}{\partial z} = 0$$

$$\frac{\partial v_z}{\partial z} = 0$$

The flow doesn't change as we move along the pipe/longitudinal direction. This is going to do something drastically nice for our problem; it's going to kill off the convective flux term we spent so much time whining about!

$$\nabla \vec{v} \cdot \vec{v} = v_z \frac{\partial v_z}{\partial z} = 0$$

That means that the Reynolds tensor for this problem is exactly 0, and we shouldn't have to worry about shocks and all of those other horrible things we saw in the previous chapter because the Navier-Stokes equation becomes linear. In fact, it becomes something almost too simple to recognize:

$$0 = -\nabla p + \mu \nabla^2 \vec{v} + \rho \vec{g}$$

The Navier-Stokes equation is telling us that, as we saw before, the pressure drop down the pipe length needs to be balanced out by a momentum loss or viceversa; but our knowledge of rheology in a fluid described by Navier-Stokes is telling us specifically how that momentum loss is related to the velocity of fluid in the pipe, giving us the connection to the velocity we needed!

Recalling that the equation we're dealing with relates vectors, we can now begin to look at what this equation tells us for the individual components of these vectors. If I looked at the component of the vectors pointing in the direction of gravity in this equation, which I'll label as the  $\hat{y}$  direction, I'd get a simple relationship that looks

very familiar:

$$0 = -\frac{\partial p}{\partial y} + \rho g$$

This is just the hydrostatic equation we saw before! That means that the pressure of a fluid in a horizontal pipe increases in the direction of gravity consistent with what you'd observe in a horizontal pipe full of standing fluid. This also has the convenient effect of ensuring that gravity doesn't affect the flow; it only affects the absolute value of the pressure, which we assumed previously as not affecting the momentum loss.

Now we can take a look at the components of the vector that are perpendicular to gravity, pointing down the length of the pipe. Applying the assumptions we listed above and doing some simplifications, we find that:

$$0 = -\frac{G_1}{\mu} + \frac{1}{r} \frac{\partial(r \frac{\partial v_z}{\partial r})}{\partial r} = \frac{1}{r} \frac{\partial v_z}{\partial r} + \frac{\partial^2 v_z}{\partial r^2}$$

Not only is this equation an ordinary differential equation—we're only taking derivatives in the radial direction  $r$ —but it's also linear in the velocity! Doing some calculus tricks to solve it, we find the general solution:

$$v_z(r) = c_1 \log(r) + c_2 + \frac{G_1 r^2}{4\mu}$$

We need  $c_1 = 0$  or the velocity will go to negative infinity at the center of the pipe, and we need the velocity at the edge of the pipe to be

zero to ensure that the velocity doesn't discontinuously change at the pipe/fluid interface—giving us a value of  $c_2$  such that we finally have our velocity for pipe flow:

$$v_z(r) = \frac{G_1(r^2 - R^2)}{4\mu}$$

$$\vec{v} = \left[ \begin{array}{cc} 0 & 0 \end{array} \quad \frac{G_1(r^2 - R^2)}{4\mu} \right]$$

This type of flow is called Hagen-Poiseuille flow, and is perhaps the most famous and well-known flow in fluid mechanics. It's parabolic in shape, is 0 at the pipe edges by definition, peaks at a value of  $v_{max} = \frac{G_1 R^2}{4\mu}$  at the center of the pipe, and has a cross-sectional average of  $\langle v \rangle = \frac{G_1 R^2}{8\mu}$ .

This finally gives us an expression for the momentum loss in a horizontal, constant cross-section pipe as a function of the velocity, by solving for the pressure drop and noting the drop in pressure is equivalent to the momentum loss from the arguments we made at the start:

$$|\vec{H}| = \frac{8\mu \langle v \rangle L}{R^2}$$

In a kinder, more just world, this would be all there is to it. Unfortunately, when researchers started directly looking at pipe flows, they noticed that everything went according to theory—until they cranked the speed up high enough, and the flow began to oscillate wildy and chaotically.

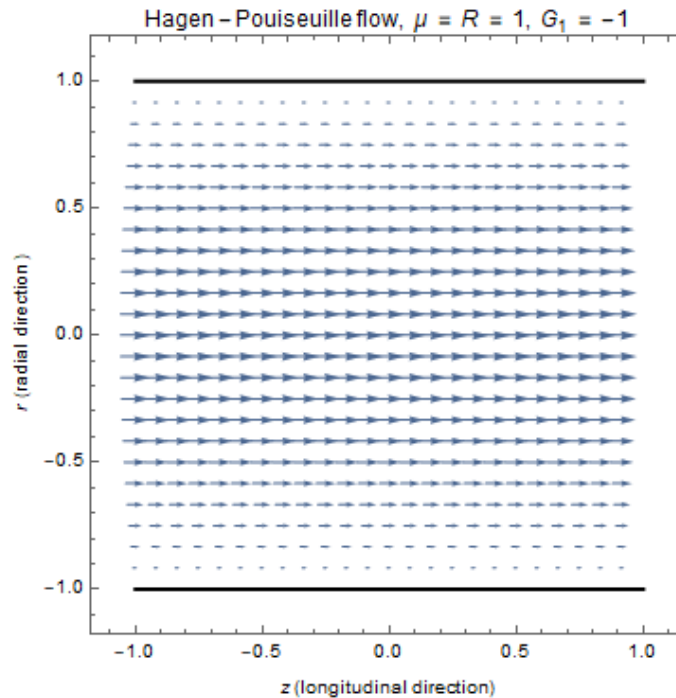


Figure 10.2: A view of Hagen-Poiseuille flow within a pipe for specific parameter values. The velocity drops to 0 at the pipe edges, denoted in black, while the velocity peaks in the center. We have assumed there is no flow in any direction other than the longitudinal one.

What the hell was going on? Well, as the existence failure for the pressure had been warning us, it *seems* as if there's another fundamental problem we didn't spot when we began to set up pipe flow; non-uniqueness! As it turns out, it appeared to experimentalists as if there wasn't just one possible flow for a given condition on a pipe inlet, but a potentially infinite number of them—and from the experiments, it seems like those other flows possess all the things we had assumed away when we derived Hagen-Poiseuille flow; they're unsteady, they have nonzero convective momentum flux, they have

nonzero flow in all three pipe directions, etcetra! These flows, which undoubtedly have nonzero Reynolds tensors, are called turbulent flows, and are omnipresent in all of fluid mechanics.

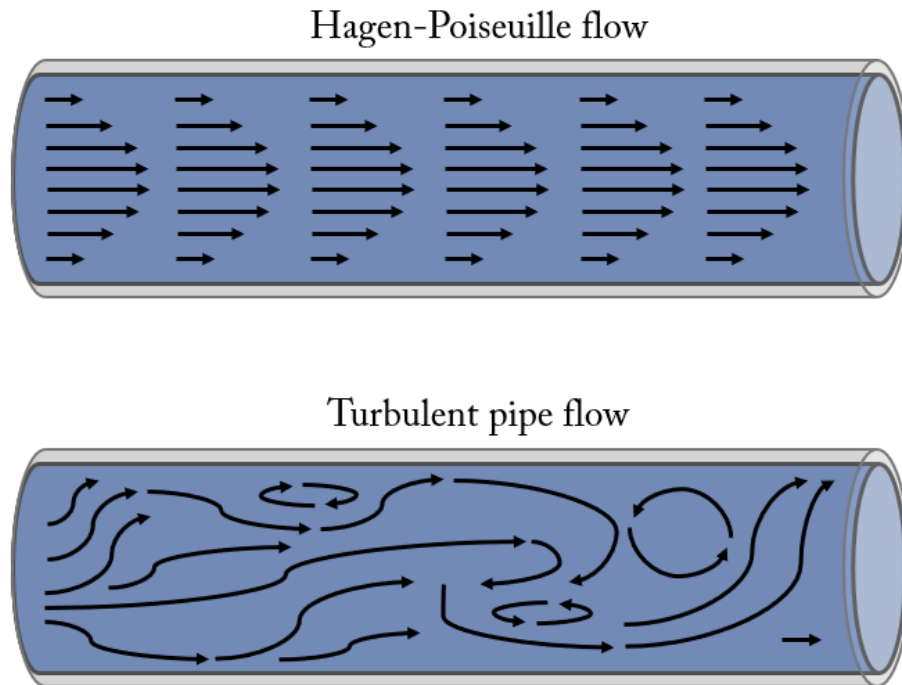


Figure 10.3: Comparison sketches between Hagen-Poiseuille and turbulent pipe flows. Hagen-Poiseuille flow is orderly and symmetric; turbulent pipe flow possesses eddies, whorls, and few if any symmetries. When the parameters of the fluid/flow are changed in specific ways, the flow transitions from one to the other.

One way fluid mechanicians explain what is happening is that when the average speed is low through the pipe, nature "picks" out the Hagen-Poiseuille flow from all other possible valid pipe flows because the litany of assumptions we made to get to Hagen-Poiseuille flow made sense; but when the average speed is sufficiently large enough, nature changes its mind and picks out one of the extremely

complicated chaotic flows instead. In mathematics, this is commonly referred to as a bifurcation. Nobody has ever been able to come up with an analytical expression for any of these alternate chaotic flows, precisely determine a theoretical criteria for this transition, or even determine whether or not this non-uniqueness hypothesis is actually true—and so we are left with experiments and numerical simulations to fill this knowledge gap for now.

But what experimentalists could confirm was curious; they observed that once a specific dimensionless number crossed a threshold, the Hagen-Poiseuille flow would switch into turbulent flow and back. That dimensionless number is the Reynolds number, which we saw in the previous chapter as an approximation to the ratio of convective to viscous effects. But that interpretation doesn't make any sense here; the convective effects in Hagen-Poiseuille flow are exactly zero, so saying that the flow transitions from laminar to turbulent because the ratio of convective to viscous effects in Hagen-Poiseuille flows becomes too large is gobbledygook. Clearly, the Reynolds number here represents something else, and to discover what it represents, we'll need to take stroll down a very curious field of mathematical physics called dimensional analysis.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. List out all the assumptions I made to justify the Hagen-Poiseuille flow. When do these assumptions make sense? When don't they?
3. In a real pipe system, what happens when the pressure drops "too much"? Do the Navier-Stokes equations apply then, and if not, what would you need to change?
4. How would you try to understand turbulence? Do you agree or disagree with the explanation above?
5. What happens if you try to include gravity? Are you able to get a solution to the Navier-Stokes equations then?
6. What happens if the pipe has a constant cross-section that isn't circular? Could you get a solution similar to Hagen-Poiseuille, and if you do, what would drive the differences?
7. Based on the results we got for pressure in the downwards and longitudinal direction, can you write a complete expression for



the pressure in a pipe section? If you want to practice changing coordinate frames, use cylindrical coordinates.

## Further Reading & Context

Nearly all introductory fluids textbooks derive Hagen-Poiseuille flow ([37][47][34][21][5][30][70][49][72][72]), and all of these discern the necessity of a linear pressure drop vs. pipe length as a consequence of the symmetry of the flow—albeit using a more concise mathematical approach.

Many fluid mechanical textbooks ([3][47][21][49][53][72], etc.) use a notation in which the expression  $\nabla \vec{v} \cdot \vec{v}$  is written as  $(\vec{v} \cdot \nabla) \vec{v}$ , with  $(\vec{v} \cdot \nabla)$  defined as an individual mathematical object sometimes referred to as the "convective operator". I avoid this, as it leads to the very confusing result that  $(\vec{v} \cdot \nabla) \vec{v} \neq \vec{v} \cdot \nabla \vec{v}$  as interpreted by standard tensor algebra.

Turbulence is, in my opinion, the greatest mystery of classical mechanics. Theories of turbulence are largely split into two often-interlinked schools of thought; the "Kolmogorov school", focusing on turbulence as a statistical phenomenon influenced by vortical energy cascades, and the "Ruelle-Takens school", focusing on turbulence as a chaotic (albeit deterministic) phenomenon described by complicated topological structures embedded in phase space. Introductions to

the former can be found in [66] and [52], while an introduction to the latter can be found in [58] or [14]. Nearly all computational efforts to understand turbulence use the Kolmogorov-type models—[52] also has a great introduction to this subject—while most analytical attempts to attack the turbulence problem use the Ruelle-Takens-type theory, which has yielded results in understanding other types of fluid-dynamical phenomena (see [17], [50], or [10]). A perspective on modeling turbulent flow properties using continuum mechanical principles within the Kolmogorov formulation, itself a very challenging and contentious subject, can be found in [29]. The Ruelle-Takens formulation of turbulence is mathematically advanced, and requires knowledge on the theory of dynamical systems to digest it properly; see [24] or [73] for an advanced introduction to the theory required to understand this formulation of turbulence. The transition to turbulence in pipe flows is still a subject of research—see [71] for an introduction to theoretical methods of investigations of this and its associated challenges.

# 11

## DIMENSIONAL ANALYSIS



*"Black Relationship", Wassily Kandinsky.*

We are, at the present, in a bit of a pickle. We have spent a lot of time deriving the laws of fluid mechanics to understand flow through a pipe, but we've found that even in something as simple as pipe flow, the inherent pathology of the laws of fluid mechanics causes pipe flow to spontaneously transition from something we can understand (Hagen-Poiseuille flow) to something we can't (turbulence). We've also commented on the fact that experimentalists have noted that the transition from one to the other occurs when the Reynolds number, which we've usually taken to represent an approximation of the ratio of convective to viscous effects in a flow, passes a critical value—even though the actual ratio of convective to viscous effects in Hagen-Poiseuille flow is identically zero. How could this be?

To solve this mystery, we're going to need apply a little bit of what may appear to be abstract nonsense. Let's again take a look at a horizontal section of pipe of radius  $R$ , with an arbitrarily long length, and a fluid for whom we prescribe a uniform pressure  $p$  and a purely longitudinal, uniform velocity with speed  $|\vec{v}|$  at the inlet. The fluid itself has a constant density  $\rho$  and constant viscosity  $\mu$ , and is under the effects of gravity with an acceleration magnitude of  $|\vec{g}|$ . These quantities should fully define the flow throughout the pipe, whatever that flow may be.

What was said above is worth repeating; every single characteristic of the flow downstream should be determined by a unique combi-



Figure 11.1: A horizontal section of pipe with a radius  $R$  and an arbitrarily long length, containing a fluid with constant density and viscosity and under the effects of gravity. The pressure and velocity are prescribed uniformly at the inlet; as the flow passes through the pipe, it will either evolve into Hagen-Poiseuille flow or turbulent flow, something fluid-mechanical theory can't tell us.

nation of these physical quantities. As a result, any physical quantity or property that we could possibly "read out" from this flow has to be a function of these 6 parameters and of these 6 parameters only.

One such readout could be some kind of turbulence index  $\zeta$ , a simple number that is 0 when the flow is Hagen-Poiseuille and 1 if the flow is turbulent. In accordance with the statement above, we can say this turbulence index  $\zeta$  is only a function of the above listed parameters:

$$\zeta = f(\rho, |\vec{v}|, |\vec{g}|, p, \mu, R)$$

Alright, this helps us out a little but not much. That being said,

we've actually constrained the problem far more than it appears we have—to demonstrate, take a look at what happens if I claimed the following:

$$\zeta = \rho$$

From a mathematical perspective, there's no problem with this, but physically, this doesn't make a lick of sense; a number like the turbulence index can't be equal to a density! This is like saying "A dog is three oranges" or "The sky is looking pretty book today"—it just doesn't possess meaning as a physical description, because we aren't relating objects that are the same type. In our case, whether or not objects are of the same type is determined by their physical dimensions. In fact, notice that it's impossible to have any expression of the form  $\zeta = f(\rho)$  make physical sense, because there's no mathematical function depending on the density that can take a density and return a simple number! And the same goes for every other physical variable that describes the system.

As a result of this, we find that we need to get a number on the right-hand side of our equality, and to get that to happen, we need to have our right hand side to be a function of just numbers as well. Now we face another question; how many independent number quantities can we construct out of those six physical parameters, and what are they? To do this, let's list out all the quantities in the system and their units (in SI):

Physical Quantity	Units (SI)
Density $\rho$	$\frac{\text{kg}}{\text{m}^3}$
Viscosity $\mu$	$\frac{\text{kg}}{\text{m s}}$
Speed $ \vec{v} $	$\frac{\text{m}}{\text{s}}$
Radius $R$	$\text{m}$
Pressure $p$	$\frac{\text{kg}}{\text{m s}^2}$
Gravitational acceleration $ \vec{g} $	$\frac{\text{m}}{\text{s}^2}$

Notice that the units of each quantity are built from multiplying and dividing three fundamental units: meters representing distance, seconds representing time, and kilograms representing mass. To get numbers with no units, it follows that we have to multiply and divide these quantities by each other to "cancel out" any and all fundamental units they may possess.

Let's start out by trying to cancel out the gravitational acceleration term  $|\vec{g}|$ . As expected, it possess units of acceleration  $\frac{\text{m}}{\text{s}^2}$ , and so we it seems natural to find quantities with units of meters, seconds, or some combination of them to try and cancel its units out. If we decided to divide  $|\vec{g}|$  by the speed squared  $|\vec{v}|^2$ , that would get rid of all of our time units; multiplying that ratio by the radius  $R$  nets us something that doesn't have units! Note that we didn't need to cancel gravity out in any specific way; I just picked out a process to try and make the unit removal as simple as possible. In any case, let's go

ahead and call this number  $\Pi_1$ :

$$\Pi_1 = \frac{|\vec{g}|R}{|\vec{v}|^2}$$

Note that we could do basically whatever we wanted to this quantity and still get a dimensionless number; we could square it, cube it, multiply it by a constant, exponentiate it, you name it; it will still be a totally valid number. In fact, most fluid mechanicians like to use the square root of the inverse of this number out of habit, which they call the Froude number  $Fr$ :

$$Fr = \sqrt{\frac{1}{\Pi_1}} = \frac{|\vec{v}|}{\sqrt{|\vec{g}|R}}$$

Alright, we've got one number down—let's try to build another one. To guarantee independence between our numbers, let's try to start the process of canceling units by picking some physical quantity that didn't show up in our numbers before, like the pressure  $p$ . It has units of force per unit area  $\frac{kg}{m s^2}$ , so we can try canceling out the time units the same way we did for  $\Pi_1$ , by dividing by the speed squared. We can then get rid of the mass units by dividing by the density, leaving us with a quantity possessing only units of distance squared, which we can readily get rid of by dividing again by the radius squared. This second number we've just found,  $\Pi_2$ , is also commonly called the Euler number  $Eu$ :

$$Eu = \Pi_2 = \frac{p}{\rho |\vec{v}|^2 R^2}$$



Neat! With these two dimensionless numbers in hand, we've utilized every physical quantity in the system except for the viscosity; it makes sense to start our search for a third number there. Viscosity has units of  $\frac{kg}{m \cdot s}$ , so we can try to divide by the speed and by the density to get rid of the time and kilogram units respectively. That leaves behind a single distance unit, which we can get rid of by dividing by the radius. This third number might look a little familiar if we explicitly show it:

$$\Pi_3 = \frac{\mu}{\rho |\vec{v}| R} = \frac{1}{Re}$$

This is just a sneaky inverted form of our good friend the Reynolds number.

Now we can ask, are there any other independent dimensionless numbers we can construct out of these quantities? As it turns out, there isn't—although there's an infinity of different numbers we can make out of these quantities, they'd all wind up being functions of the three we saw above. This is because of a famous theorem called the Buckingham pi theorem, which states that the number of dimensionless numbers one can build out of a set of physical quantities is equal to the number of physical quantities minus the number of fundamental units they're built from. (Proving this involves some relatively simple linear algebra, but it isn't really germane to what we're doing here.) In our case, we have 6 quantities and 3 units, so we get 3 dimensionless numbers as I stated before.

Given these numbers, we now have much tighter relationship between the turbulence index  $\zeta$  and the variables in the system:

$$\zeta = f(\Pi_1, \Pi_2, \Pi_3) = f^*(Fr, Eu, Re)$$

Remember that changing the specific dimensionless numbers we use as arguments doesn't really matter, whether they be  $\Pi_1$  or  $Fr$ —all the change does is alter the structure of the function, which we don't know anyways.

Nearly all of the time, fluid mechanics discard the dependence of the turbulence index on the Froude and Euler numbers by making some assumptions on the role of pressure in the flow. For example, in Hagen-Poiseuille flow, the absolute value of the pressure at either the inlet or the outlet didn't play a physical role, just their difference; if we assume this to be true for turbulent flow as well, then the flow is independent of the pressure, which means it must be independent of the Euler number. Similarly, gravity didn't play a role in the flow because all it did was generate a pressure gradient perpendicular to the flow, which doesn't factor into the equation we used to solve for the Hagen-Poiseuille flow. All it does is make the absolute pressure at the inlet non-uniform, which we've already considered irrelevant, making the turbulence index be independent on the Froude number as well.

This gives us the desired relationship that experimentalists ob-

serve:

$$\zeta = f^*(Re)$$

As a result, the Reynolds number—which we saw before as an approximation to the ratio of convective to viscous effects—is now playing a dual role as a dimensionless parameter solely responsible for determining whether pipe flow is turbulent or not. And given our predetermined form of the turbulence index  $\zeta$ , which outputs one if the flow is turbulent and 0 if the flow is Hagen-Poiseuille flow, we know everything about the function  $f^*(...)$  except for the location where the "switch" from 0 to 1 happens. Experimentalists measure the switch to happen at about  $Re \approx 2300$ .

This process that we have just described required understanding absolutely nothing about the nature of the turbulent solutions to the Navier-Stokes equation, or the nature of the process by which flow transitions from Hagen-Poiseuille to turbulent, other than some assumptions on the role of pressure. In fact, it required understanding nothing about fluid mechanics at all except for understanding the physical quantities required to uniquely define a fluid flow.<sup>1</sup> This process is called dimensional analysis, and is far more powerful than it may appear at first glance.

Consider what would happen if I as an experimentalist wanted

---

<sup>1</sup>Granted, the entire reason we had to resort to this was because we weren't finding a unique flow from this mathematical setup, but we are hypothetically observing only one type of flow for a given set of conditions, which is the flow we care about.

to understand the critical average flow speed  $|v_{\text{crit}}^{\rightarrow}|$  of some fluid through a pipe of radius  $R_0$  at which turbulence occurs, but only had access to a pipe of radius  $R_1$ . Well, because the turbulence index is only a function of Reynolds number, all I have to do is match them:

$$Re_{\text{crit}} = \frac{\rho |v_{\text{crit}}^{\rightarrow}|_1 R_1}{\mu} = \frac{\rho |v_{\text{crit}}^{\rightarrow}|_0 R_0}{\mu}$$

By doing some experiments to find the critical speed  $|v_{\text{crit}}^{\rightarrow}|_1$  in the pipe I have access to, I can then solve for the critical speed  $|v_{\text{crit}}^{\rightarrow}|_0$  I'm interested in:

$$|v_{\text{crit}}^{\rightarrow}|_0 = \frac{|v_{\text{crit}}^{\rightarrow}|_1 R_1}{R_0}$$

I can now determine the critical speed at which turbulence occurs for a given fluid flowing through a pipe of arbitrary radius by doing experiments on just one pipe. Such a set of systems is said to possess similitude, and it is a key design tool for fluid dynamicists; an engineer designing a humongous oil pipeline can rest assured that an appropriately scaled & sped-up model of the pipeline in his lab will show the same non-turbulent/turbulent behavior as his final gargantuan product. One could also change the densities and viscosities too—as long as the Reynolds number is the same, the non-turbulent/turbulent behavior will be the same.

Most engineers would call it a day here, but we're not done yet. Understanding how to trigger turbulence is one thing, but it doesn't really give us any information about the direct physical consequence

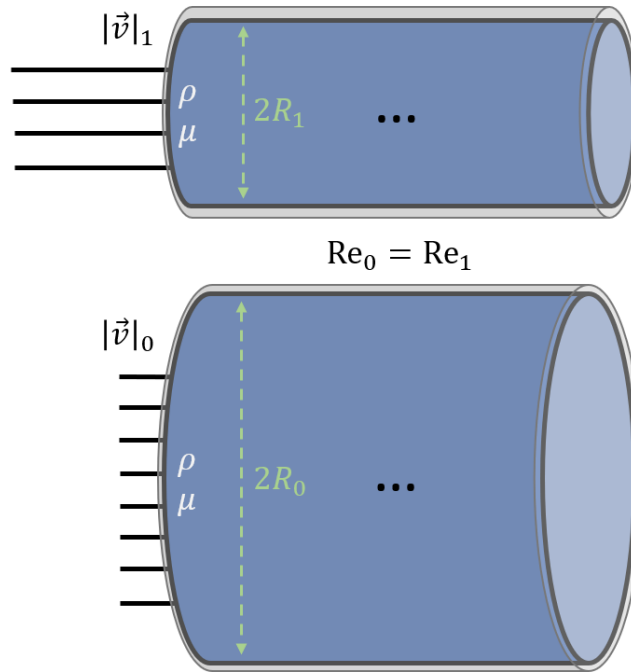


Figure 11.2: Two pipes with different radii and inlet velocities. Their inlet velocities are calibrated such that they both possess the same Reynolds number, which implies they will show the same non-turbulent/turbulent behavior. Engineers would refer to such systems as similar.

of it; how it affects the behavior of the flow in the pipe. More specifically, we don't know anything about the pressure drop in a pipe flow when the flow is turbulent. For this, we can use a slightly different flavor of dimensional analysis.

To make straightforward comparisons with the results we found before from Hagen-Poiseuille flow, we can define some variable  $\left\langle \frac{\partial p}{\partial z} \right\rangle$  representing the average pressure drop per length in a pipe flow, Hagen-Poiseuille or turbulent. This quantity has units of pressure per length, or  $\frac{kg}{m^2s^2}$ . If we ignore the roles of absolute pressure and

gravity by the arguments we justified above, we'll inevitably find an expression for the average pressure drop per unit length of the form:

$$\left\langle \frac{\partial p}{\partial z} \right\rangle = f(\rho, \mu, |\vec{v}|, R)$$

We know that dimensional analysis restricts this expression more. In fact, we know that whatever is on the right-hand side has to possess units of pressure per length, and so we should set about constructing physical quantities with units of pressure per length out of the four quantities we listed above.

To save you the trouble, there's only two such independent quantities we could construct:  $\frac{\rho|\vec{v}|^2}{R}$  and  $\frac{\mu|\vec{v}|}{R^2}$ . Consequently, one could expect that the expression representing  $\left\langle \frac{\partial p}{\partial z} \right\rangle$  could be something of the following form:

$$\left\langle \frac{\partial p}{\partial z} \right\rangle = \sum_{n \in \mathbb{R}} \alpha_n \left( \frac{\rho|\vec{v}|^2}{R} \right)^n \left( \frac{\mu|\vec{v}|}{R^2} \right)^{1-n}$$

Those  $\alpha_n$  are dimensionless, and so must be a function only of the single dimensionless quantity we can construct out of this system—the Reynolds number.

Now, recall what we said about turbulent flows before; their Reynolds tensors are decidedly non-zero. In fact, it's reasonable to expect that as we crank up the Reynolds number in a turbulent pipe flow, the Reynolds tensor is going to increase as well. Additionally, we could crank it up enough such that the convective effects

completely dominate the viscous effects! In that scenario, we'd find that all of the flow characteristics are virtually independent of the viscosity.

If that's the case, then the expression for the pressure drop per length we found above is tightly constrained by the fact that every term with the viscosity in it has to vanish, since the pressure drop per length can't depend on it. This leaves us with just the  $n = 1$  term:

$$\left\langle \frac{\partial p}{\partial z} \right\rangle = \alpha_1 \frac{\rho |\vec{v}|^2}{R}$$

We had originally stated that  $\alpha_1$ , like the other coefficients, had to be a function of just the Reynolds number. But alas—the Reynolds number is a function of the viscosity! The only way to reconcile the fact that the viscosity can't influence the physics with this Reynolds-viscosity dependence is if  $\alpha_1$  wasn't a function of anything; if it was just some arbitrary constant number, totally independent of the Reynolds number. I like to call this phenomenon, where a reduction in the number of dependent physical quantities makes it impossible to construct a dimensionless number, a dimensional crisis. Sounds cool, doesn't it?

Anyways, we've found the following result at high-Reynolds numbers for pipe flow:

$$\left\langle \frac{\partial p}{\partial z} \right\rangle_{\text{high Re}} = \alpha_1 \frac{\rho |\vec{v}|^2}{R}$$

where  $\alpha_1$  is just some constant number.

As a sanity check, we could try the same procedure when the Reynolds number is low and see if what we get matches the Hagen-Poiseuille result. In that scenario, the Reynolds tensor should be small (it is in fact identically zero), and so the viscosity-proportional term should dominate the behavior of the system. To that end, the average pressure drop per length should be independent of the only physical variable not present in the  $\frac{\mu|\vec{v}|}{R^2}$  term, the density.

That means we kill off every term in the above sum except for the  $n = 0$  term, and again trigger a dimensional crisis, leading to the following expression at the low-Reynolds number limit:

$$\left\langle \frac{\partial p}{\partial z} \right\rangle_{\text{low Re}} = \alpha_0 \frac{\mu|\vec{v}|}{R^2}$$

This precisely matches the expression we got before for Hagen-Poiseuille flow; that expression lets us determine that  $\alpha_0 = 8$ . Up to that factor of 8, we could have avoided all the differential equation-solving we did in the previous chapter by making the assumptions we did above and performing dimensional analysis; we would have gotten the same result knowing absolutely nothing about differential equations. That's pretty awesome!

I should mention that fluid mechanicians do a very cute trick when studying this average pressure drop per length; they divide the pressure drop per unit length by one of those constructed quantities with



units of pressure per length, almost always the  $\frac{\rho|\vec{v}|^2}{R}$  expression—this expression is called the Fanning friction factor  $f$ :

$$f = \frac{\left\langle \frac{\partial p}{\partial z} \right\rangle}{\frac{\rho|\vec{v}|^2}{R}}$$

The reason they do this is because now you wind up getting a dimensionless number, and we already showed that this number can only be a function of the only other dimensionless number we can construct in this system, if it's a function of anything at all—the Reynolds number. Hence, plotting the Fanning friction factor against the Reynolds number tells me the pressure drop behavior of every single possible pipe flow system of the type we describe here by virtue of similarity.

Dividing the previous results we got by this  $\frac{\rho|\vec{v}|^2}{R}$  expression, we find the following expressions for the friction factor at low and high Reynolds numbers:

$$f_{\text{low Re}} = \frac{\alpha_0 \frac{\mu|\vec{v}|}{R^2}}{\frac{\rho|\vec{v}|^2}{R}} = \alpha_0 \frac{\mu}{\rho|\vec{v}|R} = \frac{\alpha_0}{Re}$$

$$f_{\text{high Re}} = \frac{\alpha_1 \frac{\rho|\vec{v}|^2}{R}}{\frac{\rho|\vec{v}|^2}{R}} = \alpha_1$$

Having done no calculus, and relying almost exclusively on dimensional analysis, we have established reasonable hypotheses for the behavior of the Fanning friction factor as a function of the Reynolds number; in a logarithmic plot, it will show up as a straight line with

slope of -1 at the start, followed by a vertical kink somewhere due to the discontinuous transition from Hagen-Poiseuille to turbulent flow, while eventually flattening out into a straight horizontal line as the Reynolds number gets higher.

Alright then! With our prediction in hand, let's see what the experimentalists got:

Drumroll, please...

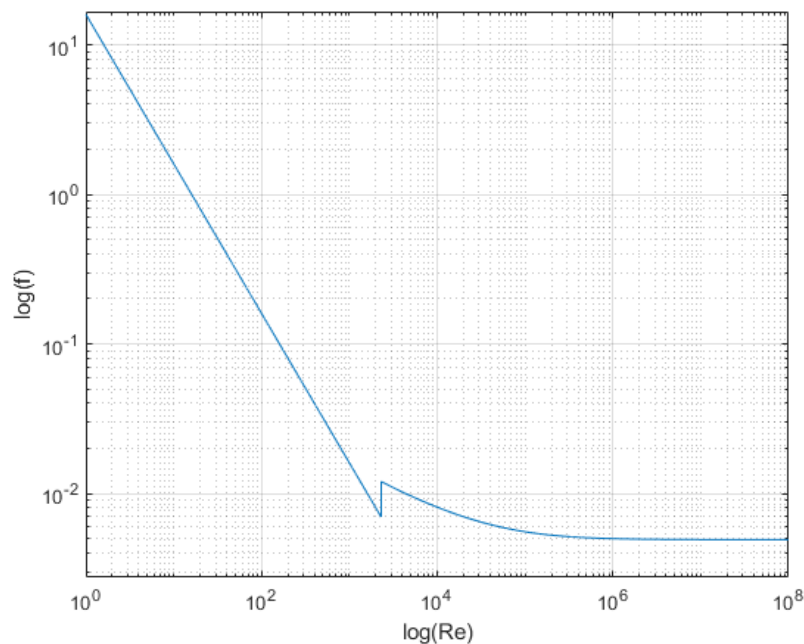


Figure 11.3: A logarithmic plot of the Fanning friction factor versus the Reynolds number in a pipe as determined by experimental data. The friction factor is proportional to the inverse of the Reynolds number for low- $Re$ , and independent of the Reynolds number at high- $Re$ . This diagram is called a Moody diagram, and fluid mechanicians often also include an empirical value called surface roughness in it to justify pipe irregularity effects. All surface roughness values lead to the same qualitative behavior as shown above.

That's one hell of a magic trick! Using the behavior at the "ends" to come close to describing the behavior in the "middle" is squarely in the ballpark of a field of math called asymptotics, and for my next trick, we're going to use it to answer the last remaining question of introductory fluid mechanics—how to characterize flow over an immersed object.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. Before we described the Reynolds number as being an approximation of the ratio of convective effects to viscous effects. How would you describe the Froude and Euler numbers in a similar way? Is there a single way to do so?
3. Try to find a physical system you don't understand and use the dimensional analysis techniques I demonstrated above to get some practical answers from them. When is it helpful? When isn't it?
4. Sometimes dimensional analysis lets us construct differential equations we can then solve. Consider dipping a hot sphere of radius  $R$  ( $m$ ) with a specific heat  $C$  ( $\frac{kg\,m^2}{s^2K}$ ) into a vat of cold liquid with a fixed ambient temperature far from the sphere. If you assume the rate of change of the temperature with time  $\frac{dT}{dt}$  is only a function of these two parameters, of the current temperature of the sphere  $T(K)$ , and of the heat transfer coefficient  $h$  ( $\frac{kg}{s^3K}$ ), find an expression for  $\frac{dT}{dt}$  using dimensional

analysis. Compare what you get to what engineers use. Try solving it, if you can! No knowledge of thermal physics is needed.

5. Physicists in the early 20th century noticed that particles at the quantum scale behave like waves. Knowing nothing about quantum mechanics other than that this wave is a function of the particle's mass  $m(kg)$ , the particle's speed  $u(\frac{m}{s})$ , and the Planck constant  $h(\frac{kg\,m^2}{s})$ , determine the wavelength ( $m$ ) and frequency ( $\frac{1}{s}$ ) of this matter wave up to a proportionality factor. Compare it to what quantum physicists theorized.
6. Do you agree with my arguments for why the Euler and Froude numbers don't matter for turbulent pipe flow? Can you think of other fluid-mechanical systems where you should have to consider them?
7. Think about how you would exploit similarity to analyze the flow of a substance like oil through a pipe versus a substance like water. What would you do?

## Further Reading & Context

Dimensional analysis is a common, ubiquitous subject in most fluid mechanics textbooks, with some textbooks introducing the notion

([33][2][34][21][49][53][72]) while others ([37][38]) rely on *a priori* knowledge of the subject to obtain meaningful dimensionless parameters. The Reynolds number is the only dimensionless parameter [37] describes, while others ([34][38][21]) discuss the Euler or Froude numbers as well, as this text does. Some textbooks ([2][47][34][21]) also discuss the Mach number—a dimensionless parameter associated with compressibility and supersonic flow—which I avoid, as this text only covers incompressible flow. In any practical application of pipe flow, there is a key parameter I chose to exclude for simplicity—the pipe surface roughness, which has been shown in experiment to play a key role in the nature of frictional losses in turbulent pipe flow. Its effect is empirically deduced, and can be seen in [48], which is also the source of the data shown in Fig. 11.3. There are countless other dimensionless parameters, such as the Strouhal or Péclet number, that emerge in specific applications of fluid mechanics.

Although a turbulence index is implicit in discussions of dimensional analysis, the naming is unique to this text—similarly, the name "dimensional crisis" for the collapse of the relationship between dimensional parameters in certain limits is introduced in this text as well. Confusingly, there is no universal scaling of the friction factor, and many texts describe friction factors that are scalar multiples of the friction factor described here ([21][34][33]). Some texts ignore the idea of friction factors entirely ([47][38][37]).

# 12

## FAR FIELD & IRROTATIONAL FLOW



*"Dawn. The Kingdom of Birds", Józef Chełmoński.*

Having said essentially everything we could about pipe flow, we now seek to characterize the last lingering question of basic fluid dynamics; flow past immersed objects.

In most descriptions of flow past immersed objects, and the only scenario we'll cover here, we have some solid, rigid, stationary object surrounded by what can be approximated as an infinite amount of fluid. These types of flows tend to be called unbounded, or are said to be in an infinite domain, as they are defined over an infinite amount of space. That doesn't necessarily mean we seek to characterize flow over all of space—just an infinite amount of it. All the way out at infinity, we'll assume that the flow is a uniform flow, with a constant velocity in a single direction and a constant pressure; the presence of an embedded object then "bends" the flow close to it in a way consistent with the Navier-Stokes equation and with conservation of mass.

This physical set-up is convenient because it is equivalent to another ubiquitous scenario in fluid mechanics, that of a rigid object moving at a constant velocity through an infinitely large medium of otherwise stationary fluid. This scenario closely approximates the dynamics of a plane or bird in the sky, or of a submarine or fish in the ocean. The correspondence between this scenario and the first one I first mentioned comes from uniformly adding or subtracting the background flow; in this latter case, the flow will be a perturbation of



a completely stationary flow by a rigid object moving with a velocity equal to the negative of the background velocity in the former case.

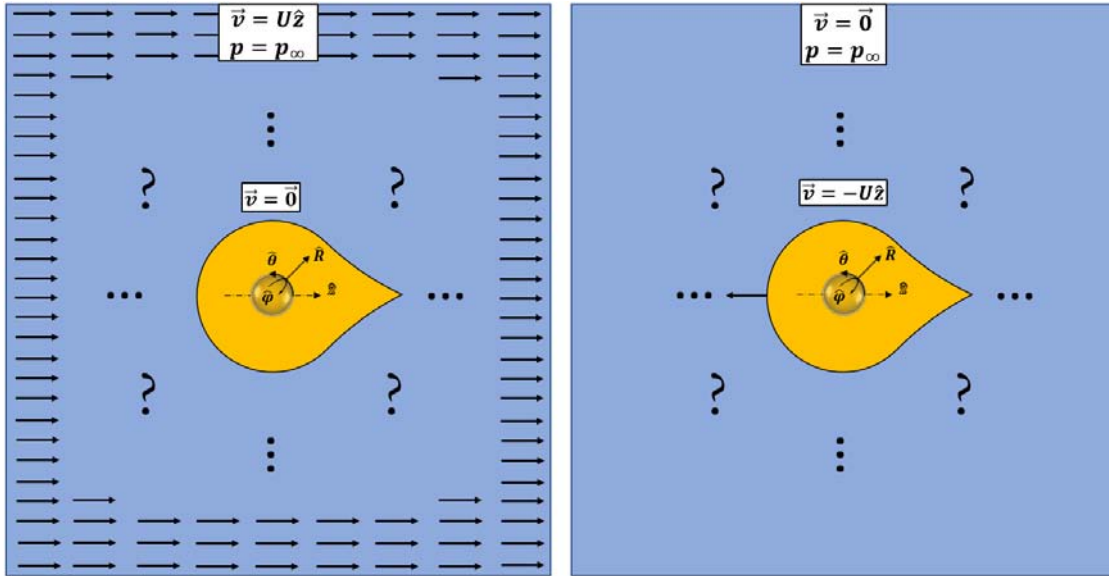


Figure 12.1: Two equivalent unbounded flows. On the left-hand side, a stationary object is placed in an infinite fluid with a uniform background flow and constant pressure at infinity. On the right-hand side, a rigid object is moving with a fixed speed and constant direction in a stationary background fluid with constant pressure at infinity. The velocity of the object in the right-hand scenario is equal and opposite to the velocity of the flow at infinity in the left-hand scenario. A spherical coordinate system at the origin has been illustrated for convenience.

With this in mind, one can consider the description of the velocity field (or pressure field, or flow) to be a combination of the background flow and of some other velocity/pressure field which decays to nothing as you get further and further away from the object. Formally, this is called a flow perturbation, in the sense that the

object "perturbs" the background flow only in a region close to the object. The velocity and pressure field would then be of the form:

$$\vec{v} = \vec{v}_b + \vec{v}_\epsilon$$

$$p = p_\infty + p_\epsilon$$

where the perturbation velocity field  $\vec{v}_\epsilon$  and perturbation pressure field  $p_\epsilon$  need to become negligibly small relative to the background velocity  $\vec{v}_b$  and pressure  $p_\infty$  respectively as we move further and further away from the immersed object.

With either set-up in mind, we could try to perform a mathematical analysis like we did for pipe flow and solve directly for the pressure and velocity everywhere. However, in this case, we don't have nearly any of the simplifying assumptions we had with pipe flow; even though we can assume the flow is steady in the stationary immersed object case, the flow can potentially be in all three coordinate directions, and it can potentially change in all three directions. In addition, we know that turbulence is baked into the Navier-Stokes equations somewhere, meaning that even the steady assumption is bound to fail at some point. What can we do?

Well, here's an idea; we could try analyzing the behavior of the flow only in a region "almost at infinity". We can't do it all the way at infinity, since all we'd get is the background flow by definition—but we can try to get a sense of what the Navier-Stokes equations and

conservation of mass tell us about the perturbation flow as we move out further and further away from the object. Formally, we are attempting to understand the asymptotic behavior of the perturbation flow near infinity, in a region commonly referred to as the far field.

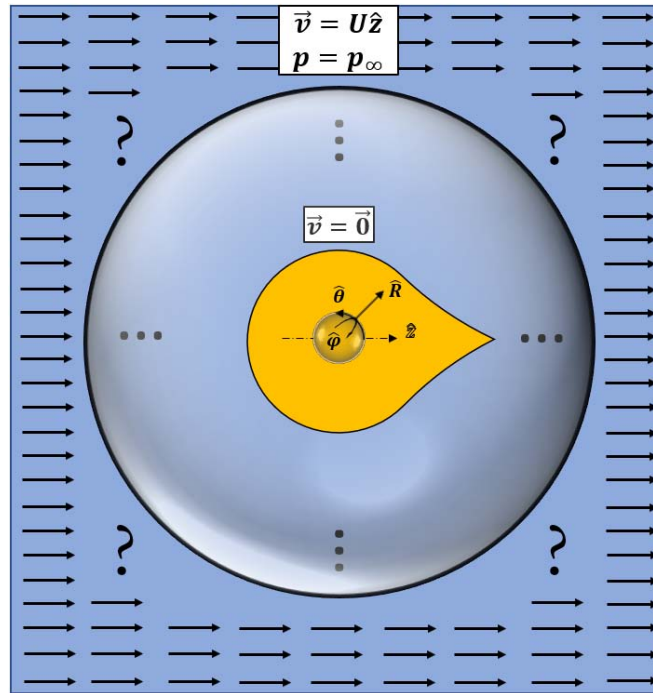


Figure 12.2: A still object is submerged in an unbounded background flow of uniform speed and constant pressure—a spherical shell around it has been constructed "almost at infinity". We want to identify the characteristics of the flow in the question mark regions outside the shell, representing the region "almost at infinity" called the far field, using asymptotic methods. A spherical coordinate system has been drawn for convenience.

Solving the full Navier-Stokes equation plus conservation of mass in this scenario isn't an option—not even the best mathematicians can do it—so what we'll do is come up with an additional assumption

about the perturbation velocity field far from the object that, when plugged in, makes the Navier-Stokes equation easier to solve. Such an assumption is commonly referred to as a kinematic constraint by dynamicists.

We want this kinematic constraint to have a couple of key features:

- It's consistent with the perturbation velocity decaying to zero as we move further and further away from the object.
- It reduces the Navier-Stokes equation to something tractable.
- It doesn't violate conservation of mass.

The idea is that now we'll have three equations we can work with; and if we make the right kinematic constraint, we'll be able to solve for the velocity or pressure straightforwardly and use the Navier-Stokes equation to calculate the fluid quantity we didn't calculate before. Because we only care about the flow outside of the boundary of the spherical shell we drew above, the flows we obtain only need to be correct in the far field; the flows we obtain from this process are far field flows.

Let's begin. By inserting the expression  $\vec{v} = \vec{v}_b + \vec{v}_\epsilon$  and  $p = p_\infty + p_\epsilon$  into the Navier-Stokes and incompressible conservation of mass equations for the scenario where we have a uniform background flow of constant speed and pressure, and under the assumption of

no external forces and steady flow, we get:

$$\rho \nabla \vec{v}_\epsilon \cdot (\vec{v}_b + \vec{v}_\epsilon) = -\nabla p_\epsilon + \mu \nabla^2 \vec{v}_\epsilon$$

$$\nabla \cdot \vec{v}_\epsilon = 0$$

The criteria for the kinematic constraint on the velocity we listed above, namely the first one regarding the flow decay at infinity, induce a very specific behavior in the perturbation flow at the far field. Informally, the idea is that the flow far from the object should be unaffected by viscous effects—only the presence of the object is triggering viscous effects in the flow, and far enough away from it, those viscous effects should be trivial. This suggests that the ratio of convective to viscous effects (i.e. the Reynolds tensor) will be large in the far field, and so  $\nabla^2 \vec{v}_\epsilon \sim 0$ <sup>1</sup>. The  $\sim$  symbol indicates "asymptotic" equivalence, i.e. that the objects are identical as one approaches the asymptotic limit, which is at  $r \rightarrow \infty$ .

This creates a problem for our strategy of calculating far field flows; we know that our biggest obstacle to a straightforward solution is that nonlinear term,  $\rho \nabla \vec{v}_\epsilon \cdot (\vec{v}_b + \vec{v}_\epsilon)$ , so it would make sense to try to get rid of it first. However, we know that in the far field,  $\nabla^2 \vec{v}_\epsilon \sim 0$ , so we can't just get rid of it outright; it would make the Navier-Stokes equation just an equation for the pressure and not the velocity in the

---

<sup>1</sup>I really wish I could prove this rigorously, but every strategy to do so that I could come up with is either pointlessly contrived or way too elaborate to show here. Oh well.

far field, which we don't want. Somehow, we have to come up with a kinematic constraint that both simplifies the nonlinear convective term and completely gets rid of the viscous term. How are we going to do that?

The secret is in that alternative form of the Navier-Stokes equations I kept showing before—the Lamb form. If I plug in my expression for the velocity, I wind up with the following representation of the Navier-Stokes equation:

$$\rho \left( \frac{\nabla (\vec{v}_\epsilon \cdot \vec{v}_\epsilon)}{2} - \vec{v}_\epsilon \times (\nabla \times \vec{v}_\epsilon) \right) = -\nabla p_\epsilon - \mu \nabla \times (\nabla \times \vec{v}_\epsilon)$$

This form makes a possible kinematic constraint obvious, one which simplifies the nonlinear terms and kills off the viscous term; the constraint that  $\nabla \times \vec{v}_\epsilon = 0$ . In general, the quantity  $\nabla \times \vec{v}$  is referred to as the vorticity  $\vec{\omega}$ , as it is proportional to the local rotation rate of the fluid. Small objects embedded in flows that satisfy this kinematic constraint do not rotate, and the flow itself is automatically unaffected by friction and rotation-related convective effects, as our Reynolds tensor calculation required for the far field.

If we take this as our constraint, we find the following system of equations:

$$\begin{aligned} \rho \left( \frac{\nabla (\vec{v} \cdot \vec{v})}{2} \right) &= -\nabla p \\ \nabla \cdot \vec{v}_\epsilon &= 0 \\ \nabla \times \vec{v}_\epsilon &= 0 \end{aligned}$$

Flows derived from this system of equations are alternatively referred to as either irrotational flows or potential flows. The latter naming convention comes from the fact that, thanks to the kinematic constraint, the velocity can be described as the gradient of a scalar quantity known as the velocity potential  $\Phi$  that satisfies the following equation:

$$\nabla^2 \Phi = 0, \quad (\vec{v} = \nabla \Phi)$$

This equation is known as Laplace's equation, and it is perhaps the easiest partial differential equation to solve (although solving it is still relatively complicated). In spherical coordinates, the general solution can be written as the sum of a bunch of different functions—to save you the trouble, here is the general solution to the Laplace equation in sum form:

$$\Phi = \sum_{\ell=0}^{\infty} \sum_{m=-\ell}^{\ell} \left( A_{\ell m} r^{\ell} + \frac{B_{\ell m}}{r^{\ell+1}} \right) Y_{\ell}^m(\theta, \varphi)$$

where the  $A_{\ell m}$  and  $B_{\ell m}$  terms represent constant coefficients, and the  $Y_{\ell}^m(\theta, \varphi)$  terms are functions of only the angles called real spherical harmonics. As either of the indices in the sum ( $\ell$  or  $m$ ) increases, these spherical harmonics become "wavier". This type of sum expression is called a multipole expansion, as the waviness causes crests, or poles, in the resulting functions.

Taking the gradient of the velocity potential to find the velocity field, we find the following general expression for the components

of all irrotational/potential flows:

$$\begin{aligned}
 v_{r_{\text{irrot.}}} &= \sum_{\ell=0}^{\infty} \sum_{m=-\ell}^{\ell} \left( A_{\ell m} \ell r^{\ell-1} - \frac{B_{\ell m} (\ell + 1)}{r^{\ell+2}} \right) Y_{\ell}^m(\theta, \varphi) \\
 v_{\theta_{\text{irrot.}}} &= \sum_{\ell=0}^{\infty} \sum_{m=-\ell}^{\ell} \left( A_{\ell m} r^{\ell-1} + \frac{B_{\ell m}}{r^{\ell+2}} \right) \frac{\partial Y_{\ell}^m(\theta, \varphi)}{\partial \theta} \\
 v_{\varphi_{\text{irrot.}}} &= \sum_{\ell=0}^{\infty} \sum_{m=-\ell}^{\ell} \left( A_{\ell m} r^{\ell-1} + \frac{B_{\ell m}}{r^{\ell+2}} \right) \frac{1}{\sin \theta} \frac{\partial Y_{\ell}^m(\theta, \varphi)}{\partial \varphi}
 \end{aligned}$$

This all just seems like useless mathematical gobbledygook. But that's because we haven't restricted our form of the potential flow by a key principle—that the velocity decay in the far field. On inspection, this immediately indicates that every  $A_{\ell \geq 1 m}$  term has to be zero, as the flow would fail to decay if they weren't thanks to the  $r^{\ell-1}$  term. The flow coming from the  $A_{00}$  term is also identically zero, as  $Y_0^0$  is just a constant that is independent of either angle and the radial part of the flow is nullified by multiplication of the  $\ell = 0$  factor. Finally, for a rigid object,  $B_{0m} = 0$ , as the flow generated by this term fails to satisfy conservation of mass in this scenario. This leaves us with:

$$\begin{aligned}
 v_{r_{\epsilon}} &= \sum_{\ell=1}^{\infty} \sum_{m=-\ell}^{\ell} -\frac{B_{\ell m} (\ell + 1)}{r^{\ell+2}} Y_{\ell}^m(\theta, \varphi) \\
 v_{\theta_{\epsilon}} &= \sum_{\ell=1}^{\infty} \sum_{m=-\ell}^{\ell} \frac{B_{\ell m}}{r^{\ell+2}} \frac{\partial Y_{\ell}^m(\theta, \varphi)}{\partial \theta} \\
 v_{\varphi_{\epsilon}} &= \sum_{\ell=1}^{\infty} \sum_{m=-\ell}^{\ell} \frac{B_{\ell m}}{r^{\ell+2}} \frac{1}{\sin \theta} \frac{\partial Y_{\ell}^m(\theta, \varphi)}{\partial \varphi}
 \end{aligned}$$

This still looks like a lot of terms. However, remember we only care about the behavior in the far field; and in the far field, no matter what



nonzero values the  $B_{\ell m}$  coefficients have, the terms that decay the slowest will always eventually become much larger than any of the other terms. As a result, asymptotically, every far field flow always looks like the flow generated by the surviving terms with the lowest radial order—which for a steadily moving rigid object, are the  $B_{1m}$  terms. Even better, the spherical harmonics corresponding to  $Y_1^m$  are really just the same function rotated 90 degrees in either angle, so we can always rotate our coordinate system to get rid of the  $Y_1^{-1}$  and  $Y_1^1$  terms. This means we get the following universal form of far field perturbation flows for rigid immersed objects moving at a constant speed:

$$\begin{aligned} v_{r_\epsilon} &\sim -\frac{2B_{10}}{r^3} \cos \theta \\ v_{\theta_\epsilon} &\sim -\frac{B_{10}}{r^3} \sin \theta \\ v_{\varphi_\epsilon} &\sim 0 \end{aligned}$$

Notice the velocity doesn't depend on  $\varphi$ ; this is because of the coordinate rotation trick I mentioned above.

I like to call this flow a dipole flow, in reference to the fact that the mathematical form of the flow field is identical to that of an electric dipole. It is important to mention that because of the analysis above, every flow perturbation caused by a steadily-moving rigid object, or by a rigid object embedded in a background flow, looks like a dipole flow in the far field. Here's what that looks like in the latter case:

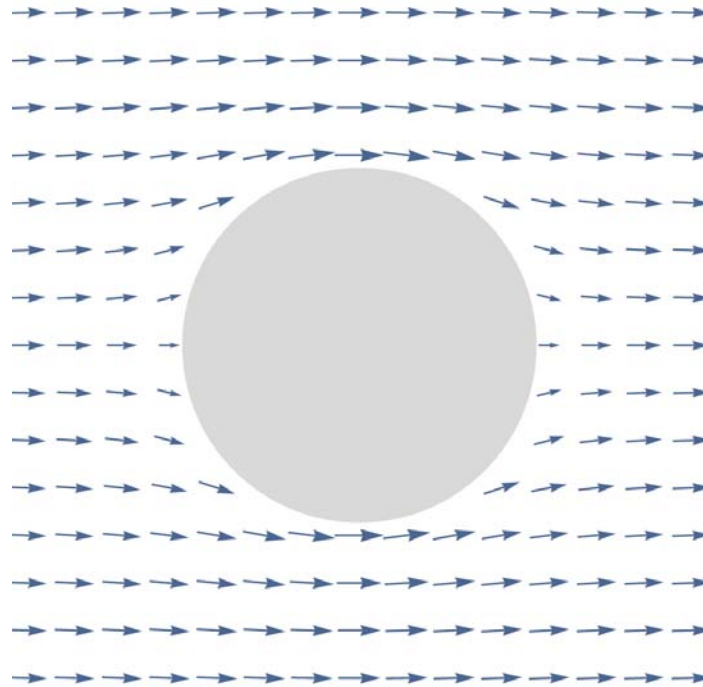


Figure 12.3: A far field flow corresponding to an object embedded in a uniform background flow. The flow is axisymmetric around the background flow axis, and is a sum of a uniform background flow and a dipole flow. The opaque grey sphere represents the near field, where the true flow behavior would be inconsistent with this flow. The background flow is deflected by the object about the background velocity axis, with the magnitude of the deflection dropping rapidly as one moves further from the object.

You might notice that the velocity vectors almost appear to be flowing over a sphere, as if the embedded object were a sphere itself. This is because every rigid object deflects flow in the far field as if it were a sphere—a mathematical consequence of the dipole flow dominating in the far field. As a result, every arbitrarily-shaped object has an effective radius  $r_{\text{eff}}$  such that a sphere with that radius

deflects flow in the far field identically to the original object.

Switching over to the rigid object moving with constant velocity in a quiescent fluid case, here's what that would look like:

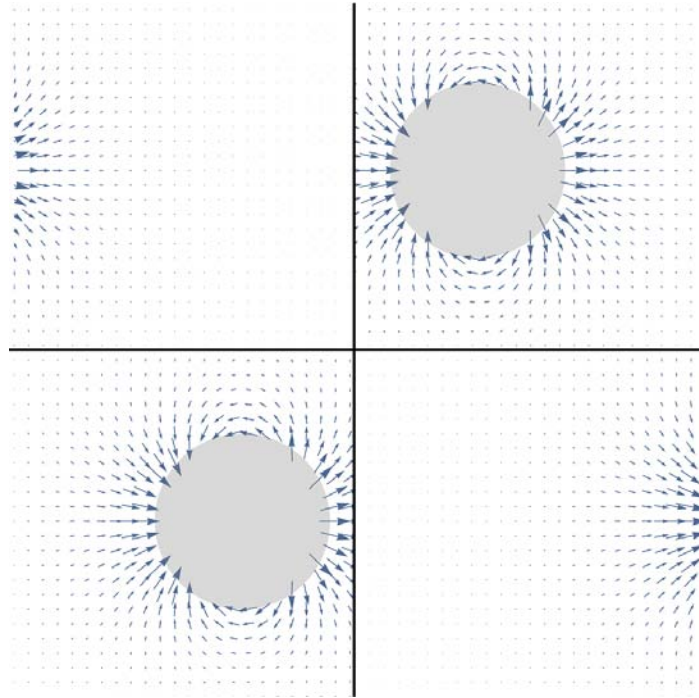


Figure 12.4: A far field flow generated by a rigid object moving with a constant speed through a quiescent fluid. The flow is an axisymmetric dipole flow with an origin moving at a uniform speed. The fluid is pushed in the direction of the object's motion in front of the object, pulled backwards in the opposite direction when near the object, and pulled forward again towards the object when behind it.

In the region immediately behind the object, the fluid is being pulled towards it; this phenomenon is called slipstreaming, and is exploited by nature and by humans to increase aerodynamic performance when traveling in groups. For example, this is one of the reasons why geese fly behind each other in V formations, or why

cyclists and race car drivers like to drive immediately behind another car when they need a speed boost. The flow patterns of all objects moving steadily while immersed in quiescent fluids behave the same way far from the objects, so it is hopefully now unsurprising that the phenomenon is ubiquitous. They can only ever be different by the scaling factor  $B_{10}$ , which is called a dipole moment in electromagnetism contexts, and a quick fly-by dimensional analysis indicates that it must be proportional to the background/object speed  $U$  times the effective radius of the object cubed  $r_{\text{eff}}^3$ .

Now that we have our expression for the perturbation velocity field, we can use the (simplified) Navier-Stokes equation to get the expression for the pressure field in the static object case:

$$\rho \left( \frac{\nabla (\vec{v} \cdot \vec{v})}{2} \right) = -\nabla p$$

It might look like we'll have to solve a partial differential equation, but notice that we're taking the gradient of a scalar in all of the terms in this equation. As a result, we can collect all the terms inside of a single gradient operation and then "integrate it out":

$$\begin{aligned} \nabla \left( p + \rho \frac{\vec{v} \cdot \vec{v}}{2} \right) &= 0 \\ p + \rho \frac{\vec{v} \cdot \vec{v}}{2} &= C \end{aligned}$$

where  $C$  is a constant.

This last equation is called Bernoulli's equation, and provides a direct algebraic relationship between the pressure and the velocity of a fluid in the case where the viscous effects are negligible, as is the case (we claim) in the far field.

Because the equation holds at every point in the far field, including at infinity, we find that Bernoulli's equation is satisfied even when the background flow is the only non-trivial flow:

$$p + \rho \frac{\vec{v} \cdot \vec{v}}{2} = p_{\infty} + \frac{\rho U^2}{2} = C$$

Noting that the total pressure is the sum of the background and perturbation pressures, and that the total velocity is the sum of the background and perturbation velocities, we can finally get an expression for the perturbation pressure in the far field:

$$p_{\epsilon} = \frac{B_{10}\rho (2r^3U(3\cos(2\theta) + 1) - B_{10}(3\cos(2\theta) + 5))}{4r^6}$$

Although it may not be immediately obvious from the expression above, this perturbation pressure is always negative, which appears to be nonsense. However, notice that the total pressure, which is the thing we need to keep positive, isn't necessarily always negative; what this result for the perturbation pressure indicates is that the effect of the object's presence on the background pressure is to decrease the pressure, more and more as one gets closer to the object. Eventually, if the other assumptions we made for the flow don't break down as

we get closer to the object, the total pressure drops so much that it must become negative—indicating that our solution for the flow can't be valid. This is yet another example of an existence failure for solutions of the Navier-Stokes equations, which we had seen before in pipe flow.

Practically, however, the liquid would boil due to the pressure reduction long before reaching that point. This occurs when the total pressure in the flow is equal to the vapor pressure  $p_{\text{vap}}$  of the fluid in question. Using the equation for the perturbation pressure above, one can solve for the distance at which the liquid would vaporize as a function of the other system parameters:

$$r_{\text{vap}}^3 = \frac{\sqrt{B_{10}^2 \rho (4(3 \cos(2\theta) + 5)p_{\text{vap}} + \rho(3U \cos(2\theta) + U)^2)}}{4p_{\text{vap}}} - \frac{B_{10} \rho U (3 \cos(2\theta) + 1)}{4p_{\text{vap}}}$$

For objects/background flows moving sufficiently fast, the pressure drop due to the presence of the object generates a vapor shell (also called a vapor cone) encasing the object, whose shape and location is described by the equation above. The equation for the vapor shell above gives us a lower estimate on what the "boundary" of the far field is, as the assumption we made when solving for far field flow clearly break down once we get inside the vapor shell due to the density change of the fluid. That being said, vapor shells rarely manifest

except in the case of exceedingly rapid objects, in which case one needs to incorporate thermodynamics into the analysis above. In most cases, viscosity begins to become relevant long before hitting the vapor shell region.

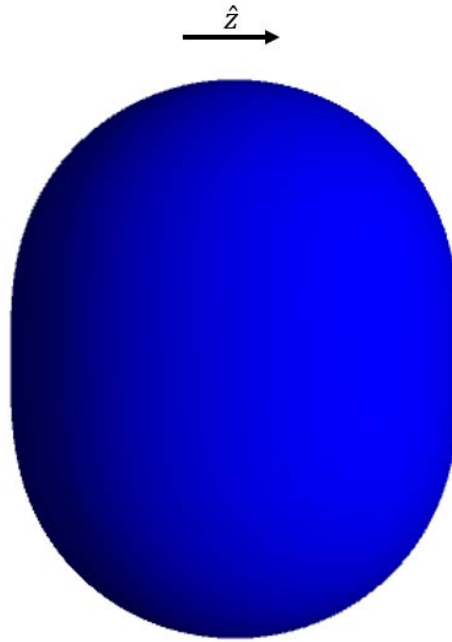


Figure 12.5: The predicted shape of the vapor shell predicted for a given set of system parameters, while ignoring pressure-temperature coupling. Changing the parameters of the system causes the shape to change. The vapor shell serves as an estimate of the boundary of the far field.

This asymptotic analysis, using the irrotational kinematic constraint on the flow field, has netted us some very nice, universal results for flow far from an object embedded in a fluid. However, as the existence failure for the pressure near the object demonstrates, it's not enough to get us some of the most important physical results

we need to design objects in fluids for engineering purposes; namely, what the force on the object from the fluid flow is, and the nature of turbulence in flows over immersed objects. For this, we'll need every trick in the book so far.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. To derive our kinematic constraint, we took the assumption that the viscous effects drop to zero in the far field. As a result, the Reynolds tensor should increase as one moves out into the far field. Is this correct, given the form of dipole flow?
3. Is the sum of two dipole flows also a potential flow? How would this be useful for calculating the flow caused by the motion of a group of distant objects, like airplanes?
4. What do you think would be the dominant behavior in the far field if we allowed the immersed object to expand/contract? You can make your life easier if you assume the expansion is really slow, so that the flow is still approximately steady.



5. Why can't we use potential flow to describe behavior in the near field, i.e. very close to the object?
6. Instead of picking the irrotational kinematic constraint, we could have also simply stated  $\nabla^2 \vec{v} = 0$ . Why didn't I do that?
7. Instead of all this math, we could have tried doing a control volume analysis over some volume enclosing the object to get all the relevant information we needed. Why wouldn't that be a good idea?
8. For the moving object case, consider a weather vane stuck on some fixed point in the far field while the object moves past it. What do you intuitively think happens to the weather vane as the objects moves past it, and how does the animation above help validate your intuition?
9. How do some of the other irrotational flows look like? Use the equations above to get a sense of what happens when you increase  $\ell$ .

## Further Reading & Context

Irrotational flow is a huge topic in introductory fluid mechanics texts, largely as a result of its analytical tractability and its ease of use in

classroom settings; see [37], [47], [33], [2], [34], [21], [38], [5], [49], [53], and [72] among many others for an introduction to these. Sometimes irrotational flow is also called inviscid or ideal flow; for incompressible fluids, irrotational flows and inviscid flows are identical. Note the presence of Bernoulli's equation in this chapter—for irrotational flow, Bernoulli's equation holds everywhere, while the "traditional" Bernoulli approximation (see the Further Reading & Context section of Chapter 5) applies only to flow along a streamline. See [33] for a discussion on the distinction. Using the irrotational flow model in regions near the object leads to inconsistent physical predictions such as d'Alembert's paradox—that irrotational flow never induces drag on an immersed object. See [5] or [38] for a discussion of this paradox.

For an introduction to the usage of asymptotics and perturbation theory in fluid mechanics, see [68] or [41], and for a general introduction into asymptotics & perturbation theory, see [6] or [55]. The name "dipole flow" comes from its mathematical equivalency to the electric field generated by a dipole—see [22] for a discussion of dipolar electric fields. See [37] and [47] for more content on spherical harmonics in fluid mechanics. The use of the vapor shell here is largely pedagogical, to demonstrate the inevitable existence failure of far field flow near the object; true vapor shells exist and occur, but are not predicted well by the equation described in this text due to the

failure of incompressibility and other thermodynamic assumptions. See [2] or [42] for a discussion of fluids where such a assumptions fail.

# 13

## NEAR FIELD & CREEPING FLOW



*"A Dew Drop Falling from a Bird's Wing Wakes Rosalie,  
who Has Been Asleep in the Shadow of a Spider's Web",  
Joan Miró.*

Using the mathematical tools of asymptotics and perturbation theory, we've managed to find what the flow past rigid stationary objects—or the flow caused by rigid moving objects—looks like far away from those objects, in the region called the far field. Now we'd like to try and see if we can pull the same tricks to get the shape of flow in the region immediately next to an object; a region perhaps unsurprisingly referred to as the near field.

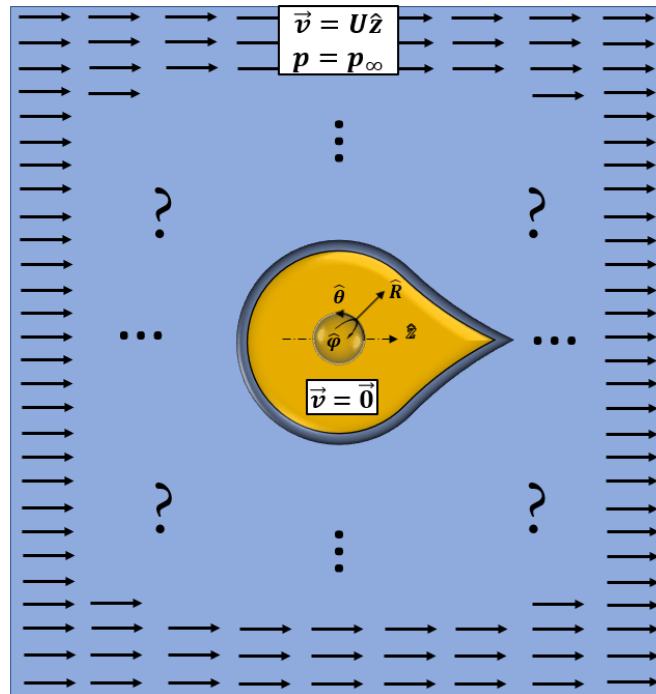


Figure 13.1: A rigid unmoving object is immersed in a uniform flow of constant pressure. We are currently interested in obtaining expressions for the flow in the region near the object, here surrounded by a transparent shell, called the near field. A spherical coordinate system has been illustrated for convenience.

One of the key reasons fluid mechanicians are interested in solving flows in the near field is to calculate forces on immersed objects.

Just as we found in Chapter 2, we'll find that the force acting on an immersed object results from integrating the tractions  $\vec{t}_s$  on the surface of the object. The difference is that now we have additional contributions to the surface traction stemming from the flow-induced molecular forces in the fluid:

$$\vec{F} = \int_A \vec{t}_s dA = \int_A (-p\hat{n} + \boldsymbol{\tau} \cdot \hat{n}) dA$$

Because these tractions are being evaluated at the surface of the object, only the characteristics of the flow in the region immediately next to the object's surface are going to influence the force on the object. Consequently, forces on objects are entirely determined by near field flow, and so we have good reason to try and characterize it.

Since we're dealing with the same physical scenario as before, that of a steady uniform flow over a still rigid object (and its moving object counterpart, by extension), we can try to use the same perturbation flow approach we used before and expand the velocities and pressures into background and perturbation parts. This time, however, the background velocity we're perturbing off of isn't the velocity at infinity—we don't care about what's happening at infinity—it's the velocity on the surface of the object.

Luckily, we have a very robust requirement for the velocity field near the object already. Our definition of fluid velocity we came up with in the very first chapter, as the weighted average of the molecules

in a tiny "molecular "net", demands that the velocity transition continuously from the velocity of the object as we move away from it. This is called the no-slip condition, and for a still object, it means that  $\vec{v} = 0$  on the surface of the immersed object. As a result, we can take our background velocity to be no velocity at all;  $\vec{v}_b = 0$ .

For the background pressure  $p_b$ , we don't really have an obvious choice. The pressure assuredly changes over the surface of the object, so there isn't a single value of the pressure at the surface that we can perturb off of. Consequently, we can simply take the background pressure to be the pressure of the flow at infinity, applied everywhere, since it's the only pressure the physical set-up we're considering is "giving" us. This has a considerable advantage in the sense that the net force across the surface of the object caused by the background pressure is identically zero, since the background pressure is the same everywhere on the surface of the object and integrating a surface normal over a closed surface is identically zero:

$$\vec{F}_{p_b} = \int_A -p_b \hat{n} dA = -p_b \int_A \hat{n} dA = 0$$

This also has the nice perk that  $\nabla p_b = 0$ , so it falls out of the perturbed Navier-Stokes equations.

Unfortunately, using the exact same physical set-up we used before for far field flow and these background quantities, we just wind up getting the original Navier-Stokes + conservation of mass system

of equations for the perturbation flow:

$$\rho \nabla \vec{v}_\epsilon \cdot (\vec{v}_b + \vec{v}_\epsilon) = -\nabla p_\epsilon + \mu \nabla^2 \vec{v}_\epsilon$$

$$\nabla \cdot \vec{v}_\epsilon = 0$$

That certainly didn't simplify things, but hopefully the following steps will.

We'd like to introduce a kinematic constraint just as we did for the far field case, but this time our constraint should be based on the behavior of the flow immediately next to the object, rather than the behavior far away from it. Because of the no-slip condition, the entire velocity field necessarily drops to zero as we move closer and closer to the object, which in this case is just the perturbation velocity field. This means that we can always define a region in which  $\rho \nabla \vec{v} \cdot \vec{v}$  is much smaller than  $\mu \nabla^2 \vec{v}$ , no matter what the values of  $\nabla \vec{v}$ ,  $\nabla^2 \vec{v}$ ,  $\rho$  and  $\mu$  are. This is equivalent to saying that the Reynolds tensor is effectively zero in this region, and it is precisely this region that we'll consider as the near field.

With this in mind, it seems pretty obvious to take as our kinematic constraint that  $\nabla \vec{v} \cdot \vec{v} = 0$ . If we do that, we'll obtain the following:

$$0 = -\nabla p_\epsilon + \mu \nabla^2 \vec{v}_\epsilon$$

$$\nabla \cdot \vec{v}_\epsilon = 0$$

$$\nabla \vec{v}_\epsilon \cdot \vec{v}_\epsilon = 0$$



This form of the Navier-Stokes equation might look familiar—it's exactly the form we had when we were solving for Hagen-Poiseuille flow. The difference is that now we don't have the unidirectionality in the flow that we had when solving for pipe flow, so we can't take this partial differential equation and turn it into an ordinary differential equation as we did before. Flows that we construct using this system of equations are usually called creeping flows or Stokes flows.

Doing a cute mathematical trick makes it really easy to solve for the pressure in a creeping flow. If you take the divergence of both sides of the Navier-Stokes equation for creeping flow, you get:

$$\nabla \cdot 0 = \nabla \cdot (-\nabla p_\epsilon + \nabla \cdot \mu \nabla^2 \vec{v}_\epsilon)$$

$$\nabla \cdot 0 = \nabla \cdot (-\nabla p_\epsilon + \mu \nabla^2 (\nabla \cdot \vec{v}_\epsilon))$$

$$\nabla^2 p_\epsilon = 0$$

That last equation for the pressure is exactly the same equation that we had for the velocity potential in far field flow, Laplace's equation! As a result, we know that the perturbation pressure has the same general expression as the velocity potential from far field flow:

$$p_\epsilon = \sum_{\ell=0}^{\infty} \sum_{m=-\ell}^{\ell} \left( A_{\ell m} r^\ell + \frac{B_{\ell m}}{r^{\ell+1}} \right) Y_\ell^m(\theta, \varphi)$$

Using the general solution to Laplace's equation in the form of spherical harmonics, you could then plug this solution for the perturbative

pressure into the (simplified) Navier-Stokes equation, and you'd get your answer for the velocity field. Easier said than done, but I digress.

However, there's a catch. The problem is that unlike irrotational flow, some Stokes flows don't necessarily cause a perturbation in the pressure! To prove it, take a look at what happens if we a priori assume that the perturbative flow has a constant perturbative pressure (including the possibility that it's zero), and so  $\nabla p_\epsilon = 0$ . With this assumption, we wind up finding:

$$0 = -\nabla p_\epsilon + \mu \nabla^2 \vec{v}_\epsilon$$

$$0 = \nabla^2 \vec{v}_\epsilon$$

$$\nabla^2 v_{\epsilon r} = \nabla^2 v_{\epsilon \theta} = \nabla^2 v_{\epsilon \varphi} = 0$$

That means that, in the case of Stokes flows with uniform perturbative pressure, each component of the perturbation velocity satisfies Laplace's equation, whose solutions have the exact same general forms as the solutions for the pressure—an infinite sum of solid harmonics. Most of the time, fluid dynamicists refer to each of these velocity fields as the homogeneous  $\vec{v}_h$  part for the uniform-pressure contribution and inhomogeneous  $\vec{v}_h$  part for the varying-pressure contribution of the near field flow:

$$\vec{v}_\epsilon = \vec{v}_h + \vec{v}_h$$

$$\nabla^2 \vec{v}_h = 0, \quad \nabla^2 \vec{v}_h = \nabla p_\epsilon$$

$$p_\epsilon(r, \theta, \varphi) = p_h + p_\mu(r, \theta, \varphi)$$

Solving both equations for each of the components of the perturbation velocity nets you the following general form for creeping flow velocity fields, which is often called Lamb's general solution. To skip you the trouble, I've written it below, where  $\vec{r}$  represents the position vector.<sup>1</sup> Be warned; this is *really* ugly.

$$\begin{aligned} \vec{v}_\epsilon = & \sum_{n=0}^{\infty} \sum_{m=-n}^n \left[ \frac{(n+3)r^2 \nabla [A_{nm} r^n Y_n^m(\theta, \varphi)]}{2\mu(n+1)(2n+3)} - \frac{n\vec{r} A_{nm} r^n Y_n^m(\theta, \varphi)}{\mu(n+1)(2n+3)} \right] + \\ & \sum_{n=1}^{\infty} \sum_{m=-n}^n \left[ (n-2)r^2 \nabla \frac{B_{nm} Y_n^m(\theta, \varphi)}{2r^{n+1}\mu n(1-2n)} + \frac{(n+1)\vec{r} B_{nm} Y_n^m(\theta, \varphi)}{r^{n+1}\mu n(2n-1)} \right] \\ & + \frac{B_{0m}\vec{r}}{2r} + \sum_{n=-\infty}^{\infty} \sum_{m=-n}^n \nabla \left[ \left( C_{nm} r^n + \frac{D_{nm}}{r^{n+1}} \right) Y_n^m(\theta, \varphi) \right] \\ & + \sum_{n=-\infty}^{\infty} \sum_{m=-n}^n \nabla \times \left[ \vec{r} \left( E_{nm} r^n + \frac{F_{nm}}{r^{n+1}} \right) Y_n^m(\theta, \varphi) \right] \end{aligned}$$

Not only does this look like mathematical logorrhea, it also contains **six** distinct infinite sequences of coefficients. However, take a look at that  $\nabla \left( C_{nm} r^n + \frac{D_{nm}}{r^{n+1}} \right) Y_n^m(\theta, \varphi)$  term inside of the third sum. This term, which is part of the homogeneous velocity field, has the same exact form as the general solution of far field flow in the previous chapter; it's the gradient of a velocity potential! As a result, the general solution for the flow velocity in the near field includes the general solution for the velocity in the far field. That doesn't mean that we can

---

<sup>1</sup>Technically, the  $B_{0m}$  term always has to be 0 since the corresponding flow fails to satisfy conservation of mass due to a mathematical quirk in vector calculus.

accurately describe far field flows using the creeping flow equations, though; flows of this form always induce pressure variations in the far field thanks to Bernoulli's equation, while the near field flows of this type never induce anything but uniform pressure changes everywhere by virtue of being part of the homogeneous velocity field.

In any case, we could try to take the general solution we got above and use the boundary conditions in the problem to restrict it into some relatively specific form we can derive conclusions about, like we did for far field flow. Unfortunately, we'll find that's not possible in general for two reasons.

The first is that the boundary condition associated with the object in the near field, that the velocity drop to zero on the surface of the object, doesn't yield a massive restriction on the coefficients in the general solution of creeping flows. This is because our solution for the flow can still go to infinity at the origin if the origin is within the object, since the flow isn't defined there anyways. That means that we can have terms that both grow and decay as we move closer to the origin, unlike what we found for far field flow (which just decays when moving closer to infinity). We can still use the condition that  $\vec{v} = 0$  on the object's surface to restrict the form of the flow, but it can only be done on a case-by-case basis as prescribed by the geometry of the object.

The second reason is associated with what near field flow should

look like as we move away from the near field—intuition might tell you that the near field flow needs to eventually look like the imposed uniform flow as we move away from the object. But this isn't necessarily true! Remember that our expressions & assumptions for near field flow are only valid in the region immediately next to the object, so expecting the near field flow to turn into the flow we prescribe at infinity is foolish in general. The near field flow should transition into something in a region somewhere between the near field and the far field, which will then turn into a dipole flow in the far field, which will then turn into a uniform flow at infinity.

That being said, sometimes constraining the near field flow by what it does at infinity isn't a terrible approximation—fluid mechanicians do it all the time, and often come up with neat mathematical tricks to make the solutions constrained by the flow at infinity more accurate. However, doing so often leads to apparent paradoxes and nonsense results, so one needs to tread lightly in case one of these paradoxes pops up in your studies.

To give you an example, take a look at two different steady flows over a rigid unmoving cylinder, when one of these flows has a larger uniform velocity at infinity:

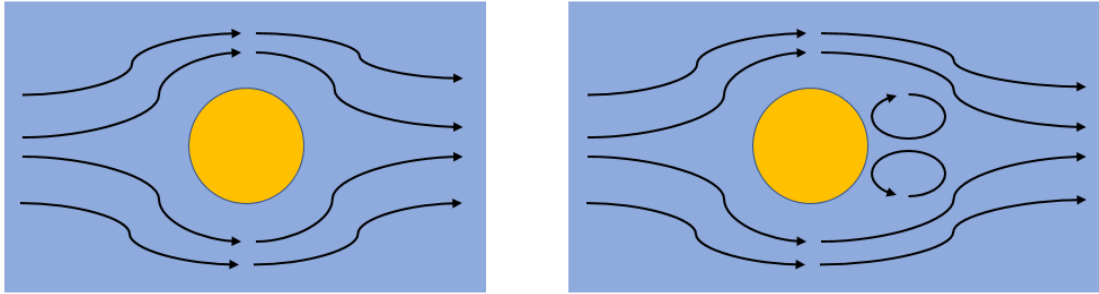


Figure 13.2: Sketches of two experimentally observed flows past a sphere; the flow on the right has a larger background velocity. The flow on the left matches the Stokes prediction everywhere in the near field using the boundary condition that the perturbative flow decay at infinity. The flow on the right does not, even in the near field region.

The solution one finds from the general form of creeping flows, combined with applying boundary conditions based on the uniform flow at infinity, matches the flow with no vortices almost identically everywhere in the flow. However, it doesn't really seem to match the flow with the vortices; which would be fine, if the flows were dissimilar only after some distance away from the object. However, the flows don't match even in the region we'd think they should, right next to the object where we said that flows are always creeping flows. How could this be happening?

The reason is that the flow in that region that isn't quite the near or the far field is different, which means that the boundary conditions for the creeping flow in the near field change! So the flow immediately next to either sphere is still a creeping flow, they're just different

creeping flows because the boundary conditions for the creeping flow induced by the flow in the not-quite-near/not-quite-far region are different. I haven't seen this mentioned in other fluid mechanics books, but I like to call it the matching paradox.

All of this should be motivation enough to dig into what flows look like in the not-quite-near, not-quite-far region. Formally, this region is often called the boundary layer, and it contains perhaps the most notable feature of flows past rigid immersed objects; wakes.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. How would you estimate some characteristic length representing how far the near field extends from the object?
3. Given the form of  $\boldsymbol{\tau}$  we found before, what does  $\boldsymbol{\tau} \cdot \hat{n}$  look like as a function of the velocity? What does it convey physically about the flow-induced forces on the object?
4. What other choices for the background pressure could you make, and how would they be better or worse than the choice we made?

5. How are the assumptions that we made to get the creeping flow equations here different from the assumptions we made to get the same equations in pipe flow?
6. What parts of the perturbation pressure induce a net force on an immersed objects?
7. When solving for creeping flow over a sphere, one can find a unique solution that both matches the boundary conditions both at infinity and at the surface of the object. Why is this solution only valid in the near field, given what we know about the flow behavior in the far field?

## Further Reading & Context

In most fluid mechanics texts, near field flows are not discussed as Stokes/creeping flows; they are discussed as boundary layer flows, which use different, more restrictive kinematic assumptions (see the Further Reading & Context section of Chapter 14). That being said, Stokes/creeping flows are an important part of fluid mechanics—and are usually used to model flows that are either "very" slow or at the microscale. In these texts, the Stokes flow approximation is used over the entire flow field—near and far—producing useful models but leading to matching paradoxes such as the Stokes paradox and the



Whitehead paradox. The usage of the phrase "matching paradox" is non-standard and is introduced in this text—it encompasses not just paradoxes arising from near field flow, but also from far field flow, such as d'Alembert's paradox (see the Further Reading & Context section at the end of Chapter 12). See [37], [5], [47], [34], or [32] for an introduction to this usage of Stokes flows, and see [26], [31], [59] or [23] for more advanced specific texts on using Stokes/creeping flow models at the microscale. The reader may find Lamb's general solution in [37] or [26], albeit not written as explicitly as it is here. Further elaboration on the mathematics of Stokes/creeping flows can be found in [41]. Experimental observations of the recirculation zone sketched in Figure 13.2, and when it begins to form as a function of a Reynolds number, can be found in [65]. The no-slip condition here is justified as a result of averaging arguments based on the continuum approximation, which is unconventional—more detailed discussions on the physical phenomenology of the no-slip condition, and different rationalizations for it, can be found in [32] and [40].

# 14

## BOUNDARY LAYERS, LIFT, & DRAG



*"Airplane Synchromy in Yellow-Orange",  
Stanton Macdonald-Wright.*

Now that we've said everything we can about flows far from immersed objects and flows near immersed objects, it makes sense to try and understand the flow everywhere in between. Even though we won't be able to make too many quantitative statements about it—the math is way too complicated in that region—we will be able to make a couple of general statements about it.

The term boundary layer is often used as a catch-all term for any region between the surface of the object and the background flow at infinity, but here we use it specifically to refer to the regions in the fluid that are neither described by near or far field flow assumptions. Most of the time, this region is illustrated or described as a velocity field that monotonically increases in magnitude from 0 at the surface of the object to the background speed. Such a description is both shockingly common and always incorrect.

Although this can be proved rigorously, it suffices to look at the form of far field flow we derived two chapters ago; the velocity in the far field increases as we get closer to the object. As a result, the flow speed can't just increase slowly until it gets to the background flow speed as we move away from the object. It needs to increase in the near field, and then keep increasing until it gets to a speed that's larger than the background speed somewhere within the boundary layer, and then decay into the background speed in a way consistent with our expressions for far field flow. There isn't a consistent name

for this, but I like to call it the speed bump.

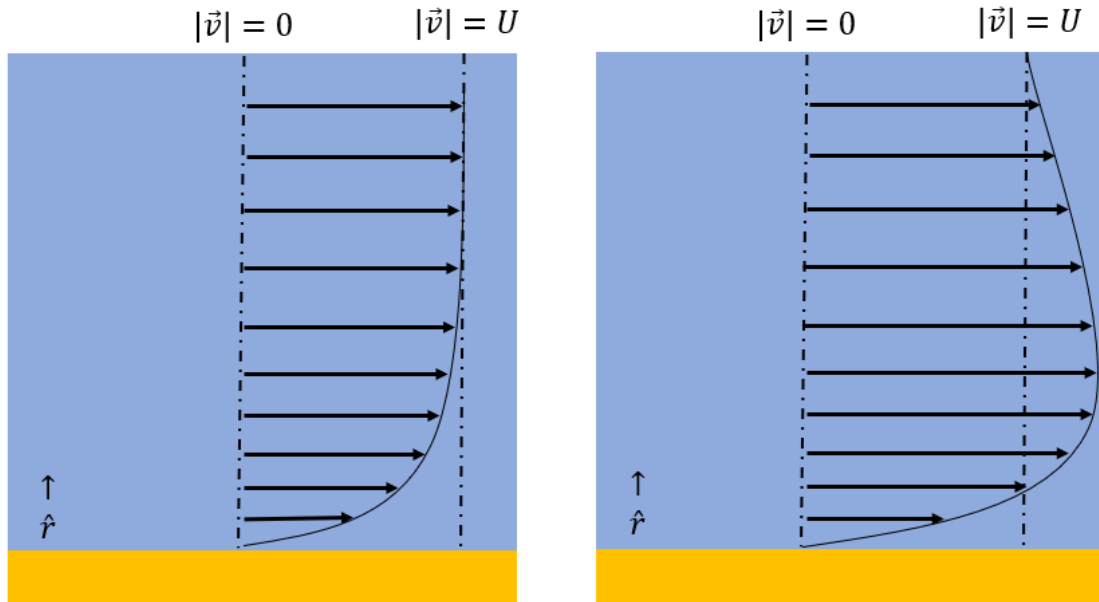


Figure 14.1: Two sketches of velocity profiles in the boundary layer. The common representation on the left assumes that the flow speed is always something between 0 (at the object surface) and the background flow speed  $U$  (at infinity). The more accurate representation on the right demonstrates the flow speed "overshooting" the background flow speed and then decaying into the background flow speed  $U$  at infinity, consistent with our findings for far field flow. This is the so-called "speed bump".

Another qualitative feature of boundary layer flow that plays an important role in fluid dynamics is that it can reverse directions. In such a scenario, the near field flow around the object is pointing in the opposite direction as the far field flow associated with the object/flow, and so the flow needs to "turn around" somewhere in the boundary layer—this was the case for the sphere with a vortex pair

behind it that we saw in the last chapter, for example. Because the velocity changes continuously, as the flow "turns around", there has to be some point at which the flow velocity becomes zero when moving away from the object. Around these points, the fluid effectively behaves as if it were near the surface of an imaginary object, and so any fluid bounded by the object and these imaginary surfaces effectively becomes locked in within it. This phenomenon is called flow separation, as the fluid flow appears to "separate" off of the object, and the flow bounded within the zero-velocity surfaces is said to be recirculating flow. Unsurprisingly, the region of fluid bounded by these surfaces is called the recirculation zone. This is by-and-large all we can generically say about steady flow in the boundary layer. That being said, we're going to find that—just as in pipe flow—flow in the boundary layer always becomes turbulent and unsteady once certain conditions are met, and we can find those conditions using the same technique we did for pipe flow; dimensional analysis.

To make things a little easier, let's take a look at our tried-and-true scenario of a fixed rigid object immersed in a fluid with density  $\rho$  and viscosity  $\mu$ , with a background speed  $U$  and background pressure  $p_\infty$ . But this time, we'll specifically make the immersed object a sphere with radius  $R$ . These parameters define everything about the flow around the sphere.

As we saw for pipe flow, the Navier-Stokes equation (and by ex-

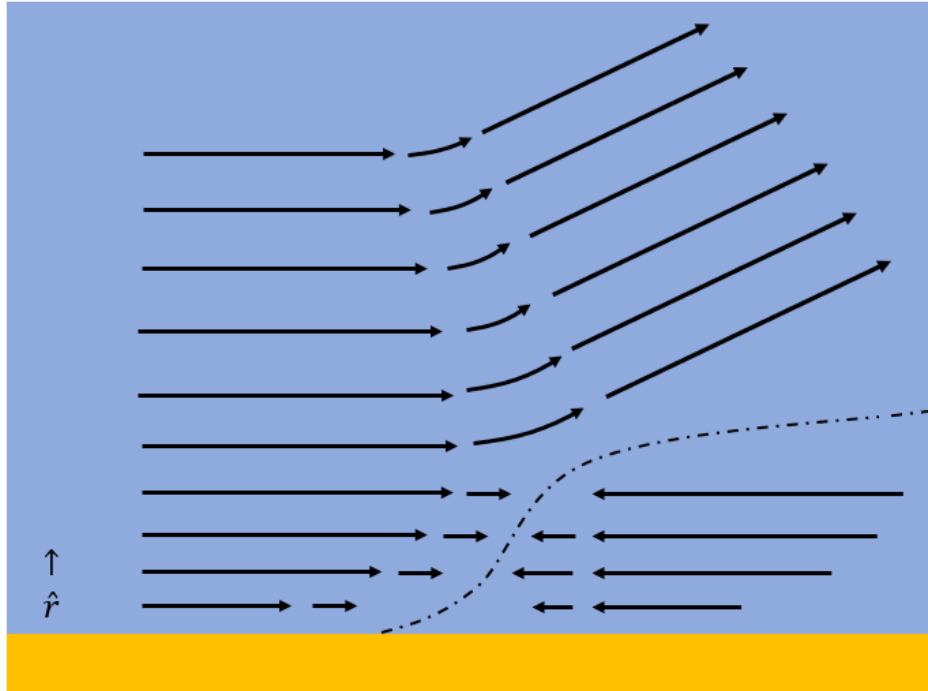


Figure 14.2: A sketch of the process of flow separation. Flow near the object is pointing in opposite directions, indicating that there is a region (the dotted line) where the flow velocity is zero within the fluid. Flow bounded from above by the dotted line lies within the recirculation zone, and fluid within this zone cannot leave it.

tension the physics of this problem) only involves pressure gradients, not the absolute pressure. And because the total pressure in the system is the sum of an absolute background pressure that is uniform everywhere and a perturbation pressure that isn't, none of the pressure gradients are affected by the value of the background pressure! Consequently, the system is independent of the background pressure as long as the pressure isn't so low that the liquid boils.

That leaves us with 4 dimensional parameters that define the flow based on 3 units of measurement—length, mass, and time. If we

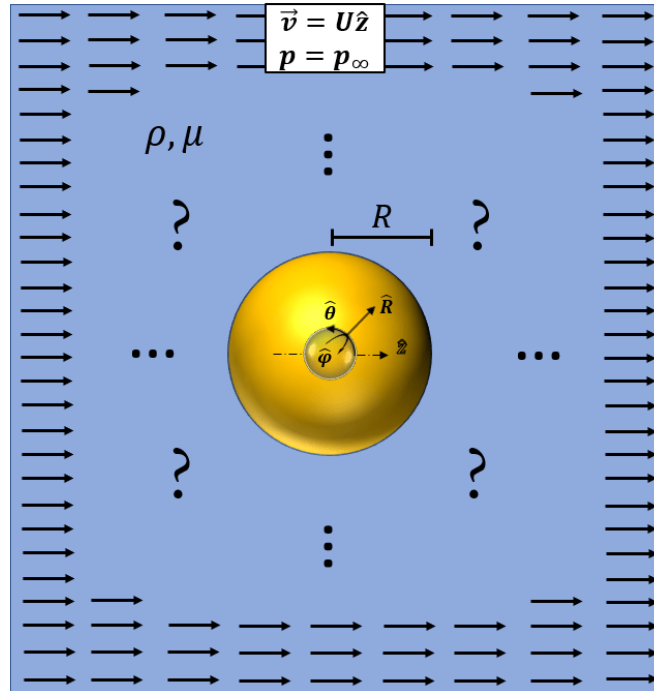


Figure 14.3: Steady flow past a sphere of radius  $R$ . The fluid has a uniform density  $\rho$  and viscosity  $\mu$ . At an infinite distance from the object, the flow is uniform with speed  $U$  and pressure  $p_\infty$ . This fully defines the flow everywhere.

tried to construct a turbulence index  $\zeta$  as a function of these four parameters, we'd find that (thanks to the Buckingham Pi theorem) the turbulence index can only be a function of a single dimensionless number—the Reynolds number  $Re$ .

$$\zeta = f(Re) = f\left(\frac{\rho U R}{\mu}\right)$$

Just like we saw in pipes, whether or not the flow past an immersed sphere is steady or turbulent is entirely decided by the Reynolds number. But the transition to turbulence in flow past spheres (and immersed objects in general, for that matter) has a very rich struc-

ture that doesn't just go from "nice and clean" to "chaotic and unpredictable". This structure is largely present in all flow transitions for flows past immersed objects.

For steady flow past a sphere, the flow takes one of two forms—flow with no recirculation zone at the lowest Reynolds numbers, or flow with a small recirculation zone in the back consisting of two counter-rotating vortices at the small-but-not-smallest Reynolds numbers. When the Reynolds number increases enough, we find that the flow

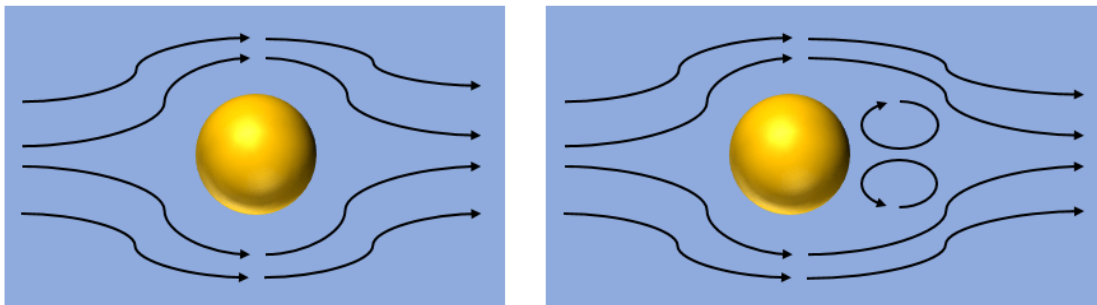


Figure 14.4: Two forms of steady flow past an immersed sphere. For the smallest Reynolds numbers, the flow does not recirculate, and the flow lines are symmetric in the "front" and "back" of the sphere. As the Reynolds number increases, a recirculation zone (or wake) forms behind the sphere, composed of two counter-rotating vortices.

fails to be steady, but in a peculiar way—the vortices in the recirculation zone begin to wiggle perpendicular to the flow, and "detach" from the object, forming a streak of vortices called a von Karman vortex street. These emitted vortices travel downstream many distances longer than the radius of the sphere, and are the beginning of what we might recognize in common parlance as a wake. Although this



flow is turbulent, it is quite simple and structured, and can be studied theoretically (although we will not do so here). As the Reynolds

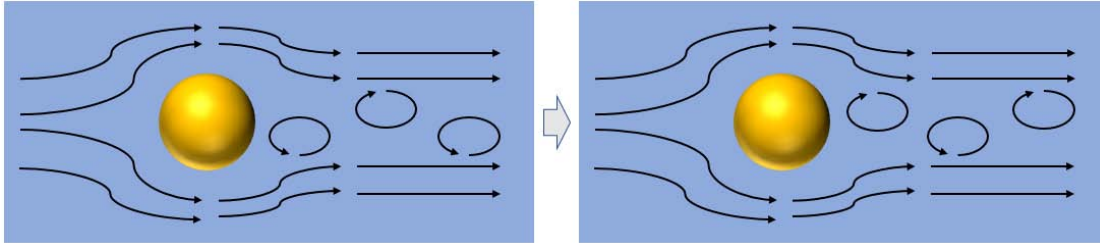


Figure 14.5: Flow past an immersed sphere at Reynolds numbers where flow begins to become unsteady. This unsteadiness begins to manifest as a von Karman vortex street, which is an unsteady, periodic downstream "emission" of counter-rotating vortices off of the back of the sphere. This forms a much larger recirculation zone than in the steady flow case.

number increases further, the structure of the vortices being emitted from the back of the object decompose into a turbulent chaotic mess, and the flow appears to look well-behaved everywhere except for in a long messy "tail" immediately behind the object—this is a proper wake. Finally, increasing the Reynolds number even more results in the flow becoming turbulent everywhere in the boundary layer, not just in the wake. In this final stage, the flow has become fully turbulent and behaves very much as it does in turbulent pipe flow, where it possesses very little structure or order anywhere. These last two stages of turbulence are commonly referred to as subcritical and supercritical, respectively.

Given that dimensional analysis gave us a way to at least classify

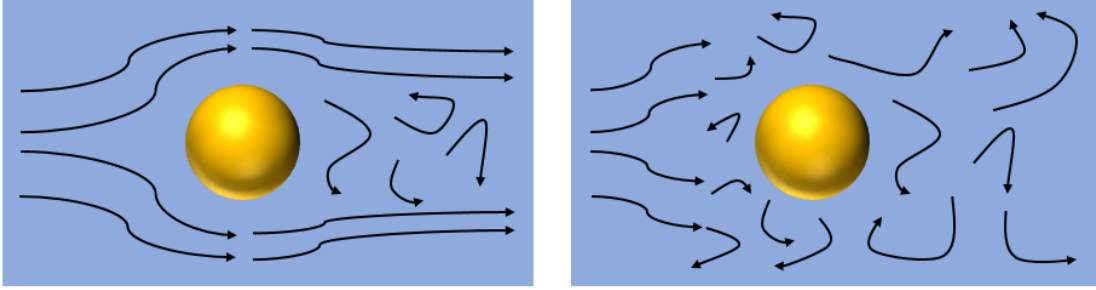


Figure 14.6: Subcritical (left) and supercritical (right) turbulent flow past an immersed sphere. In subcritical flow, the flow behind the sphere possesses a large recirculation zone (or wake) filled with chaotic unsteady flow, but the flow outside of this zone is steady. In supercritical turbulent flow, the flow everywhere in the boundary layer is turbulent.

observed flows past an immersed sphere, perhaps it will be useful to answer another extremely important question about flow past immersed objects; the forces that fluids exert on them.

Usually, the forces on an immersed object are split up into components that are parallel and perpendicular to the background flow direction—the parallel component is usually called the drag force, and the perpendicular component is called lift force. Drag and lift are both just different components of the same force, caused by both pressure-induced tractions and flow-induced molecular interaction tractions on the surface of the object. The general equation for the force on an immersed object is then:

$$\vec{F} = \int_A \vec{t}_s dA = \int_S -p\hat{n} + \boldsymbol{\tau} \cdot \hat{n} dA$$

$$\vec{F} = \int_A -p_\epsilon \hat{n} + \mu \left( \nabla \vec{v}_\epsilon + \nabla \vec{v}_\epsilon^T \right) \cdot \hat{n} dA$$

Notice the similarity to the expression we found in Chapter 2 for buoyant forces on a submerged object; this formula is a generalized version of it accounting for fluid flow.

However, we can't solve for the flow in the boundary layer, and so we don't really have a chance to derive anything for the net force acting on the object and its lift/drag components using this expression. But we can use dimensional analysis, coupled with some situational assumptions, to get surprisingly general expressions.

For example, we can try to find how many independent things with units of force we can construct out of the set of 4 dimensional parameters we have available for flow past a sphere. It turns out we can only make two,  $\mu U R$  and  $\rho U^2 R^2$ . This means that either the lift or the drag force on a sphere can be represented in the following form:

$$F_d \text{ or } F_l = \sum_{n \in \mathbb{R}} \alpha_n \left( \rho U^2 R^2 \right)^n (\mu U R)^{1-n}$$

This is correct, but essentially useless; we need some other assumptions to get a workable, specific expression. One assumption we could make is that the surface tractions coming from the viscous effects are much smaller than those coming from the pressure perturbations; mathematically, that  $\mu |(\nabla \vec{v} + \nabla \vec{v}^T)| \cdot \hat{n} \ll |p| \hat{n}$  on the surface of the sphere. This is true if the viscosity is very small, or if the average velocity gradient on the surface of the sphere is on average zero because the flow in the boundary layer is chaotically

flipping direction. In either scenario, the Reynolds number would be very large.

In this scenario, the drag/lift force is independent of the viscosity, and so anything dependent of it must have no bearing on the drag/lift force—namely, the  $\mu U R$  term. As a result, for high Reynolds numbers, we expect the drag and lift to have the following form:

$$F_d \text{ or } F_l = \alpha_1 \rho U^2 R^2$$

where  $\alpha_1$  is a numerical constant independent of any of the other physical parameters in the system, a consequence of the dimensional crisis caused by removing the viscosity. This kind of drag, which is quadratic in the velocity, is usually called Newtonian drag.

Alternatively, we can consider what happens when the viscosity is very large, or when the Reynolds number is very low. In such a scenario, the terms dependent on the viscosity in the above sum should dominate, and anything independent of the viscosity shouldn't influence the drag force; particularly, the  $\rho U^2 R^2$  term. Removing the only parameter that solely appears in this expression, the density, we find the following expression for the drag/lift force at low Reynolds numbers:

$$F_d \text{ or } F_l = \alpha_0 \mu U R$$

where  $\alpha_0$  is also just a number. This kind of drag is called Stokes drag, and it is linear in the velocity of the object/background flow. If one

attempts to solve for Stokes flow over a sphere assuming the near field flow equations apply everywhere (which is incorrect), one would find that  $\alpha_0 = 6\pi$  for the drag force relationship.

Usually, fluid mechanics plot the drag (or lift) force in terms of a nondimensional number called a drag (or lift) coefficient  $C_d$  or  $C_l$ , which is the drag force divided by  $\rho U^2 R^2$  or some multiple of it:

$$C_d \text{ or } C_l = \frac{F_d \text{ or } F_l}{\rho U^2 R^2}$$

This nondimensionalization of the drag/lift force is convenient because  $C_d$  or  $C_l$  can only be a function of nondimensional parameters constructed from the physical variables that describe flow past a sphere. And there's only one nondimensional parameter we can make with those four variables; the Reynolds number.

If you were to plot the drag coefficient versus the Reynolds number  $Re$  for a sphere based on experimental results, this is what you'd find:

Drumroll again, please...

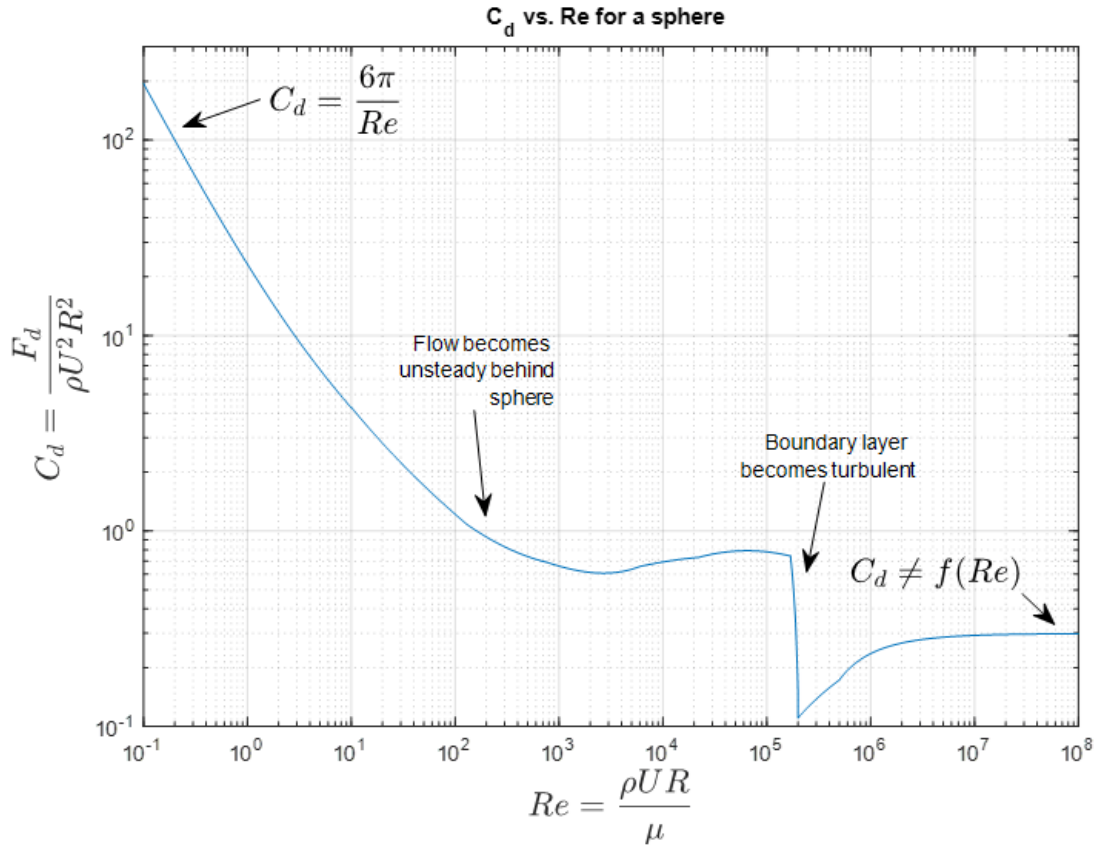


Figure 14.7: The drag coefficient as a function of the Reynolds number for a sphere, determined by experiment. At low  $Re$ , the relationship between the drag force & speed is linear, resulting in an inversely proportional relationship between the drag coefficient and the Reynolds number. At high  $Re$ , the drag force is quadratic in the speed, leading to the drag coefficient being independent of the Reynolds number at high  $Re$ .

This is all consistent with what we expected from dimensional analysis! Now we might ask, how do any of these results change when the object isn't a sphere, but something totally different?

A simple way to illustrate this might be by considering what would happen if I put a spherical "nose" on the sphere with a radius  $R_2$ , di-

rectly in the front of the sphere. This parameter  $R_2$  defines everything about the nose, and  $R$  along with  $R_2$  fully define the geometry of this new class of immersed object.

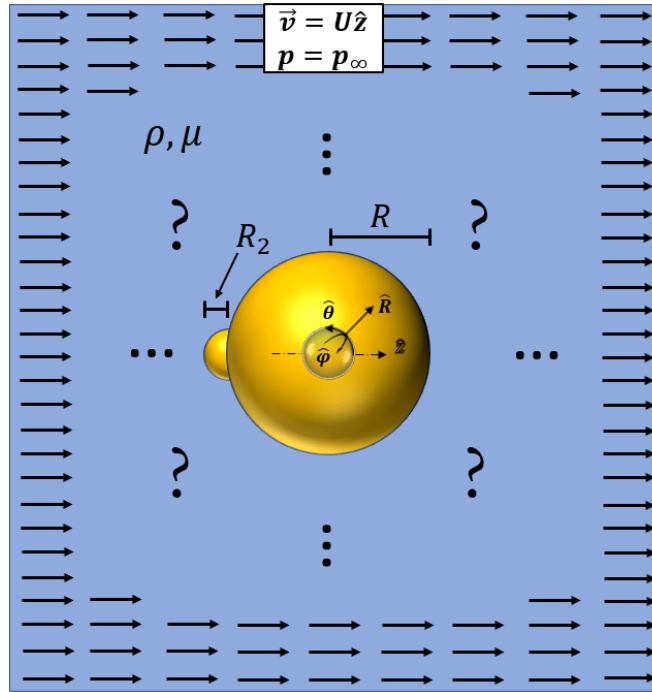


Figure 14.8: Flow past an immersed object consisting of a sphere of radius  $R$  with a "nose" of radius  $R_2$ . The flow in this system is uniquely defined by the same parameters as for the sphere, with the addition of the "nose" radius.

If I did the whole dimensional analysis rigmarole I did for the sphere for this new object, I'd find the same things except for the presence of another, new dimensionless parameter:  $\frac{R_2}{R}$ . More importantly, if I tried to find expressions for the drag/lift force in the limits of low and high Reynolds numbers, I'd get stuck with a bunch of horrid sums again:

$$F_d \text{ or } F_l \text{ for low Re} = \sum_{n \in \mathbb{R}} \alpha_n (\mu U R)^n (\mu U R_2)^{1-n}$$

$$F_d \text{ or } F_l \text{ for high Re} = \sum_{n \in \mathbb{R}} \alpha_n (\rho U^2 R^2)^n (\rho U^2 R_2^2)^{1-n}$$

However, in either case, the  $\alpha$ 's can only be a function of the only dimensionless objects we can construct; the radii ratio  $\frac{R_2}{R}$ . Because of that, we can actually factor out a couple of things out of the sum! Simplifying a bit, we'll find that everything that isn't a function of either radius winds up on the outside:

$$F_d \text{ or } F_l \text{ for low Re} = \mu U \sum_{n \in \mathbb{R}} \alpha_n R^n R_2^{1-n}$$

$$F_d \text{ or } F_l \text{ for high Re} = \rho U^2 \sum_{n \in \mathbb{R}} \alpha_n R^{2n} R_2^{2^{1-n}}$$

Everything that's within the sums is only a function of the geometry of the object, and so can be grouped into a geometric factor  $\Gamma$  which we choose to have units of length. With this, we find that:

$$F_d \text{ or } F_l \text{ for low Re} = \mu U \Gamma_{\text{low}}$$

$$F_d \text{ or } F_l \text{ for high Re} = \rho U^2 \Gamma_{\text{high}}^2$$

The great thing about this is that, if we were to add another geometric feature to the immersed object (say, "ears"), we'd wind up finding the exact same thing, albeit with a presumably more complicated



geometric factor involving more geometric parameters. As a result, one can construct whatever kind of immersed object one wants using features, each defined by a single length, and the above results will still hold! For example, this indicates that at the very low and very high Reynolds number limits, the drag/lift force dependence on the viscosity, density, and speed are independent of the shape of the object. Anywhere in between, the geometry of the object does affect those relationships by affecting the values of the  $\alpha_n$ 's.

## Things To Think About

1. How does the artwork displayed in the beginning of the chapter relate to the concepts described here?
2. What do you think affects the magnitude of the speed bump, and why? Do you expect to be bigger at lower Reynolds numbers or higher ones?
3. What physical phenomena do you think affect flow separation? How could you encourage or ameliorate it?
4. Do you think that altering the geometry of an object can remove some of the "stages" of flow between non-recirculating and fully turbulent? How or why not?

5. Why do you think the drag coefficient for a sphere sharply drops after the flow becomes supercritically turbulent?
6. Approximating a swimmer as a funny-looking sphere, does a swimmer experience Stokes drag, Newtonian drag, or something in between?

## Further Reading & Context

The boundary layer is often defined as a region that includes the near-field, and is described in that scenario using a model that includes both viscous and inertial effects, albeit in a highly simplified way. Often, the assumption is that the flow there is two-dimensional, laminar & steady among other things—which leads to the derived assumption that such a boundary layer does not perturb the pressure field, and that a speed bump does not occur. An introduction to this perspective can be found in many introductory fluid dynamics textbooks ([37][47][33][21][2][34]). A comprehensive and voluminous elaboration of the subject of boundary layers can be found in [61]. Real images captured of the emergence of unsteady phenomena in flows past immersed objects can be found in [69] and [60]. With the exception of scaling factors in lift/drag coefficients and the idea of a geometric factors, which is not discussed in a general way in other introductory texts to my knowledge, lift & drag on an immersed

body are covered here in a way essentially equivalent to that in other texts ([47][33][21][5][2][34][53][72][49]). The graph shown in Fig.14.7 is the "standard drag curve", constructed from a dozen experimental studies of flow past a sphere—see [11] for its form & detailed list of sources. For actual calculated examples of geometric factors and their influence in the flow of asymetrically objects in a microhydrodynamic context, see [31].

# BIBLIOGRAPHY

- [1] M. P. Allen and D. J. Tildesley. *Computer Simulation of Liquids*. Oxford University Press, 2017.
- [2] J. D. Anderson. *Fundamentals of Aerodynamics*. McGraw-Hill Series in Aeronautical and Aerospace Engineering. McGraw-Hill, 1991.
- [3] R. Aris. *Vectors, Tensors and the Basic Equations of Fluid Mechanics*. Dover Books on Mathematics. Dover Publications, 2012.
- [4] H. A. Barnes, J. F. Hutton, and K. Walters. *An Introduction to Rheology*. Rheology Series. Elsevier Science, 1989.
- [5] G. K. Batchelor. *An Introduction to Fluid Dynamics*. Cambridge University Press, 1973.
- [6] C. M. Bender and S. A. Orszag. *Advanced Mathematical Methods for Scientists and Engineers I: Asymptotic Methods and Perturbation Theory*. Springer, 1999.
- [7] R. B. Bird, W. E. Stewart, and E. N. Lightfoot. *Transport Phenomena*. Wiley, 2006.
- [8] H. D. Block. *Introduction To Tensor Analysis*. C.E. Merrill Books, 1962.
- [9] A. I. Borisenko and I. E. Tarapov. *Vector and Tensor Analysis with Applications*. Dover Books on Mathematics. Dover Publications, 1968.
- [10] P. Chossat and G. Iooss. *The Couette-Taylor Problem*. Applied Mathematical Sciences. Springer New York, 2012.

- [11] R. Clift, J. R. Grace, and M. E. Weber. *Bubbles, Drops, and Particles*. Dover Civil and Mechanical Engineering. Dover Publications, 2013.
- [12] C. F. Colebrook. Turbulent flow in pipes, with particular reference to the transition region between the smooth and rough pipe laws. *Journal of the Institution of Civil Engineers*, 11(4):133–156, 1939.
- [13] A. A. Collyer and D. W. Clegg. *Rheological Measurement*. Springer Netherlands, 2013.
- [14] P. Cvitanović, R. Artuso, R. Mainieri, G. Tanner, and G. Vattay. *Chaos: Classical and Quantum*. ChaosBook.org, Niels Bohr Institute, Copenhagen, 2020.
- [15] S. R. De Groot and P. Mazur. *Non-Equilibrium Thermodynamics*. Dover Books on Physics. Dover Publications, 2013.
- [16] W. M. Deen. *Analysis of Transport Phenomena*. Topics in Chemical Engineering. Oxford University Press, 2012.
- [17] P. G. Drazin and W. H. Reid. *Hydrodynamic Stability*. Cambridge Monographs on Mechanics. Cambridge University Press, 1981.
- [18] D. J. Evans and G. P. Morriss. *Statistical Mechanics of Nonequilibrium Liquids*. DOAB Directory of Open Access Books. ANU E Press, 2007.
- [19] R. P. Feynman, R. B. Leighton, and M. Sands. *The Feynman Lectures on Physics, Vol. II: Mainly Electromagnetism and Matter*. Feynman Lectures on Physics. Basic Books, 2011.
- [20] E. M. Furst and T. M. Squires. *Microrheology*. Oxford University Press, 2017.
- [21] R. A. Granger. *Fluid Mechanics*. Dover Publications, 1995.
- [22] D. J. Griffiths. *Introduction to Electrodynamics*. Cambridge University Press, 2017.
- [23] É. Guazzelli, J. F. Morris, and S. Pic. *A Physical Introduction to Suspension Dynamics*. Cambridge Texts in Applied Mathematics. Cambridge University Press, 2011.
- [24] J. Guckenheimer and P. Holmes. *Nonlinear Oscillations, Dynamical Systems, and Bifurcations of Vector Fields*. Applied Mathematical Sciences. Springer New York, 2013.
- [25] J. P. Hansen and I. R. McDonald. *Theory of Simple Liquids*. Elsevier Science, 2006.

- [26] J. Happel and H. Brenner. *Low Reynolds Number Hydrodynamics: With Special Applications To Particulate Media*. Mechanics of Fluids and Transport Processes. Springer Netherlands, 2012.
- [27] K. Huang. *Statistical Mechanics*. Wiley, 1987.
- [28] J. H. Hubbard and B. H. West. *Differential Equations: A Dynamical Systems Approach: Ordinary Differential Equations*. Texts in Applied Mathematics. Springer New York, 2013.
- [29] K. Hutter and K. Jöhnk. *Continuum Methods of Physical Modeling: Continuum Mechanics, Dimensional Analysis, Turbulence*. Springer, 2004.
- [30] W. S. Janna. *Introduction to Fluid Mechanics*. Taylor & Francis, 2009.
- [31] S. Kim and S. J. Karrila. *Microhydrodynamics: Principles and Selected Applications*. Dover Civil and Mechanical Engineering. Dover Publications, 2013.
- [32] B. J. Kirby. *Micro- and Nanoscale Fluid Mechanics: Transport in Microfluidic Devices*. Cambridge University Press, 2013.
- [33] B. J. Kirby. *How Fluids Work*. Independently published, 2019.
- [34] P. K. Kundu, I. M. Cohen, and D. R. Dowling. *Fluid Mechanics*. Elsevier Science, 2015.
- [35] O. A. Ladyzhenskaya. Solution “in the large” to the boundary value problem for the navier-stokes equations in two space variables. In *Doklady Akademii Nauk*, number 3, pages 427–429. Russian Academy of Sciences, 1958.
- [36] O. A. Ladyzhenskaya and R. A. Silverman. *The Mathematical Theory of Viscous Incompressible Flow*. Martino Publishing, 2014.
- [37] H. Lamb. *Hydrodynamics*. Dover Books on Physics. Dover Publications, 1945.
- [38] L. D. Landau and E. M. Lifshitz. *Fluid Mechanics*, volume 6 of *Course of Theoretical Physics*. Elsevier Science, 2013.
- [39] R. G. Larson. *The Structure and Rheology of Complex Fluids*. Topics in Chemical Engineering. OUP USA, 1999.
- [40] E. Lauga, M. Brenner, and H. Stone. *Microfluidics: The No-Slip Boundary Condition*, pages 1219–1240. Springer Berlin Heidelberg, 2007.

- [41] L. G. Leal. *Advanced Transport Phenomena: Fluid Mechanics and Convective Transport Processes*. Cambridge Series in Chemical Engineering. Cambridge University Press, 2007.
- [42] H. W. Liepmann and A. Roshko. *Elements of Gas Dynamics*. Dover Books on Aeronautical Engineering. Dover Publications, 2013.
- [43] E. Madelung. Quantentheorie in hydrodynamischer form. *Zeitschrift für Physik*, 40(3–4):322–326, 1927.
- [44] J. E. Marsden and A. Tromba. *Vector Calculus*. W. H. Freeman, 2003.
- [45] G. T. Mase, R. E. Smelser, and G. E. Mase. *Continuum Mechanics for Engineers*. Computational Mechanics and Applied Analysis. CRC Press, 2009.
- [46] D. A. McQuarrie. *Statistical Mechanics*. University Science Books, 2000.
- [47] L. M. Milne-Thomson. *Theoretical Hydrodynamics*. Dover Books on Physics. Dover Publications, 2013.
- [48] L. F. Moody. Friction factors for pipe flow. *Trans. ASME*, 66:671–684, 1944.
- [49] B. R. Munson, D. F. Young, T. H. Okiishi, and W. W. Huebsch. *Fundamentals of Fluid Mechanics*. Wiley, 2009.
- [50] J. M. Ottino. *The Kinematics of Mixing: Stretching, Chaos, and Transport*. Cambridge Texts in Applied Mathematics. Cambridge University Press, 1989.
- [51] R. L. Panton. *Incompressible Flow*. Wiley, 2013.
- [52] S. B. Pope. *Turbulent Flows*. Cambridge University Press, 2000.
- [53] P. J. Pritchard and J. W. Mitchell. *Fox and McDonald's Introduction to Fluid Mechanics*. Wiley, 2015.
- [54] R. F. Probstein. *Physicochemical Hydrodynamics: An Introduction*. Wiley, 2005.
- [55] R. H. Rand and D. Armbruster. *Perturbation Methods, Bifurcation Theory and Computer Algebra*. Springer New York, 2012.
- [56] P. Renteln. *Manifolds, Tensors, and Forms: An Introduction for Mathematicians and Physicists*. Cambridge University Press, 2013.

- [57] A. Rodriguez-Gonzalez, J. P. Gleghorn, and B. J. Kirby. Rational design protocols for size-based particle sorting microdevices using symmetry-induced cyclical dynamics. *Phys. Rev. E*, 101:032125, 2020.
- [58] D. Ruelle and F. Takens. On the nature of turbulence. *Communications in Mathematical Physics*, 20(3):167–192, Sep 1971.
- [59] W. B. Russel, D. A. Saville, and W. R. Schowalter. *Colloidal Dispersions*. Cambridge Monographs on Mechanics. Cambridge University Press, 1991.
- [60] M. Samimy, K. S. Breuer, L. G. Leal, and P. H. Steen. *A Gallery of Fluid Motion*. Cambridge University Press, 2004.
- [61] H. Schlichting and K. Gersten. *Boundary-Layer Theory*. Springer Berlin Heidelberg, 2016.
- [62] D. V. Schroeder. *An Introduction to Thermal Physics*. Addison Wesley, 1999.
- [63] G. Segré and A. Silberberg. Radial particle displacements in poiseuille flow of suspensions. *Nature*, 189(4760):209–210, January 1961.
- [64] A. J. M. Spencer. *Continuum Mechanics*. Dover Books on Physics. Dover Publications, 2012.
- [65] S. Taneda. Experimental investigation of the wake behind a sphere at low reynolds numbers. *Journal of the Physical Society of Japan*, 11(10):1104–1108, 1956.
- [66] H. Tennekes and J.L. Lumley. *A First Course in Turbulence*. MIT Press, 1972.
- [67] C. Truesdell. The mechanical foundations of elasticity and fluid dynamics. *Journal of Rational Mechanics and Analysis*, 1:225–231, 1952.
- [68] M. Van Dyke. *Perturbation Methods In Fluid Mechanics*. Number 8 in Applied Mathematics and Mechanics. Academic Press, 1964.
- [69] M. Van Dyke. *An Album of Fluid Motion*. Parabolic Press, 1982.
- [70] Z. Warhaft. *An Introduction to Thermal-Fluid Engineering: The Engine and the Atmosphere*. Cambridge University Press, 1997.
- [71] F. M. White. *Viscous Fluid Flow*. McGraw-Hill Education, 2005.
- [72] F. M. White. *Fluid Mechanics*. McGraw-Hill, 2011.



- [73] S. Wiggins. *Introduction to Applied Nonlinear Dynamical Systems and Chaos*. Texts in Applied Mathematics. Springer New York, 2003.