

Introduction to Fluid Dynamics

Joseph D. MacMillan



This version was compiled on October 2, 2022. For the most up-to-date version
and supplementary material, see
josephmacmillan.github.io/IntroductionToFluidDynamics.

This work is licensed under a Creative Commons Attribution-ShareAlike 4.0
International License.

Contents

Contents	ii
Preface	vi
Who is this book for?	vi
What does this book cover?	vii
What else do you need to teach a course in fluid dynamics?	viii
Why Open Education?	ix
Thanks	ix
1 Introduction	1
1.1 Fluids	1
1.2 Describing Fluid Flow	2
1.3 Visualizing Fluid Flow	4
1.3.1 Vector Plots	4
1.3.2 Streamlines	4
1.4 Total Derivative and Acceleration	6
1.5 Incompressible and Irrotational Flow	8
1.5.1 The Incompressibility Condition	8
1.5.2 Vorticity	10
1.6 Viscosity	13
Problems	15
2 Viscous Fluids	17
2.1 The Navier-Stokes Equations	17
2.1.1 Cartesian Coordinates	18
2.2 Simple Viscous Flows	18
2.2.1 Poiseuille Flow in One Dimension	18
2.2.2 Time Dependent Flow	21
2.2.3 An Impulsively Moved Boundary	26
2.2.4 Self-Similar Flows	29
2.3 Circular Flow	36

2.3.1	The Navier-Stokes Equations in Cylindrical Coor-	
dines		36
2.3.2	Uniformly Rotating Fluid	37
2.3.3	Spin Down of Rotating Fluid	39
2.3.4	The Line Vortex	44
2.4	Transport of Vorticity	47
	Problems	49
3	Ideal Fluids	53
3.1	Describing Ideal Flow	53
3.1.1	Static Fluids	53
3.1.2	Bernoulli's Principle	54
3.1.3	The Vorticity Equation	56
3.1.4	Circulation	58
3.1.5	The Surface of a Rotating Fluid	60
3.2	The Velocity Potential and Stream Function	62
3.2.1	The Velocity Potential	62
3.2.2	Flow Past a Cylinder	64
3.2.3	The Stream Function	69
	Problems	72
4	Potential Flow	76
4.1	The Complex Potential	76
4.1.1	The Complex Plane	77
4.1.2	Basic Flows	79
4.1.3	Method of Images	83
4.1.4	The Circle Theorem	87
4.2	Flow Past Objects	90
4.2.1	Flow Past a Cylinder	90
4.2.2	Conformal Mapping	93
4.2.3	Flow Past an Elliptical Cylinder	96
4.2.4	Flow Past a Flat Plate	98
4.2.5	Flow Past an Aerofoil	100
4.3	Vortex Motion	103
4.3.1	The Helmholtz Vortex Theorems	103
4.3.2	A Vortex Pair	106
4.3.3	Vortex Shedding	109
4.3.4	The von Kármán Vortex Street	112
	Problems	117
5	Waves	121
5.1	Gravity and Capillary Waves	121

5.1.1	Surface Waves	121
5.1.2	The Kinematic and Pressure Conditions	123
5.1.3	Small Amplitude Waves	125
5.1.4	Sinusoidal Waves on Deep Water	126
5.1.5	Capillary Waves	130
5.2	Non-Sinusoidal Waves	134
5.2.1	Fourier Series	134
5.2.2	Wave Packets	137
5.2.3	Group Velocity and Energy	140
5.2.4	Long-Time Response	143
5.3	Sound Waves	147
5.3.1	Compressible Fluids	147
5.3.2	Small Amplitude Sound Waves	149
	Problems	153
6	Special Topics	155
6.1	Deriving the Navier-Stokes Equation	155
6.1.1	The Stress Tensor and Vector	155
6.1.2	Cauchy's Equation of Motion	157
6.1.3	Newtonian Fluids	159
6.2	Boundary Layers	162
6.3	Nonlinear Waves	166
6.3.1	The Shallow Water Equations	166
6.3.2	Method of Characteristics	168
6.3.3	The Dam Break Problem	170
6.4	Flow Past a Sphere	175
6.4.1	Ideal Flow	175
6.4.2	Stokes Stream Function	178
6.4.3	Very Viscous Flow	179
6.4.4	Slow Flow Past a Sphere	180
	The Equation of Motion	181
	The Boundary Conditions	182
	The Solution	183
6.4.5	Stokes Law	185
	The Pressure	185
	The Stress	187
	The Force on the Sphere	188
6.5	Stellar Structure	189
6.5.1	Newtonian Gravity	189
6.5.2	Energy Transport	190
6.5.3	Equations of Stellar Structure	191
6.5.4	Polytropic Models	194

6.6 Computational Methods	197
6.6.1 Computational Fluid Dynamics	199
6.6.2 Smoothed Particle Hydrodynamics	200
6.6.3 Lattice Boltzmann Method	202
Problems	204

Preface

I have to confess right away that I'm not an expert in the subject of fluid dynamics; I was given an opportunity to teach a course on it a number of years ago and didn't want to refuse, and I assumed my single graduate-level course in fluids was enough preparation. I'm an astrophysicist by trade, but not one who normally does astrophysical fluid research. Regardless, I threw myself into the course, and ended up teaching it numerous times, improving my notes each time. I initially taught from Acheson's very fine *Elementary Fluid Dynamics*, but found it a little too advanced for my students, who were upper undergraduate physics and math students. Other fluid dynamics textbooks I found were either graduate-level like Acheson's or geared towards engineering and applied fluid mechanics. So, as all textbooks are created as far as I'm aware, I resolved to turn my binder of notes into a textbook to fit the gap: I wanted a clear, concise, and readable book aimed at upper undergrads in physics and math. I started it in 2014 or so, but didn't really make any progress until 2022 when I finally had time, in the form of a development leave, to focus on it. It was a tremendous amount of fun to write.

Who is this book for?

This is for undergraduate physics and math students interested in solving problems in fluid dynamics. I assume some basic background – you, the reader, should have some experience with vector calculus and solving ordinary differential equations – but even then I'll remind you when necessary what you probably learned in your second year physics and math classes. Otherwise, I introduce the math when we need it: there's a short section on complex numbers, I spend quite a bit of time going through the process of separation of variables to solve partial difference equations, and topics like Fourier analysis is introduced in the amount we need.

On the other hand, graduate students in physics or math likely won't find enough here to keep them interested beyond the basics – there's no real discussion of instability or turbulence, or shock waves, or gas dynamics, or other advanced topics, and the amount of time spent on computational methods is much too brief. Similarly, engineering students likely won't find enough applied work here; the focus is consistently on theory and problem-solving, and many common engineering topics typical in fluid mechanics courses is missing.

To keep things familiar for undergrads, I've used notation from Griffiths' very popular *Introduction to Electrodynamics*, even when not commonly in use in the subject (I use s for the radial coordinate in the cylindrical coordinate system, for example). And I've tried to keep a fairly informal tone throughout the book, so I hope students find it easy to read and not too dense.

What does this book cover?

The basics of fluid flow as I see them. That means, after a brief introduction on notation, terms, and visualization, starting with the famous Navier-Stokes equation. Most students at this point in their career have at least heard of it (and how hard it is to solve!), and solving the first example (Poiseuille flow) is a great introduction to PDEs. But there's only so much you can do analytically with the Navier-Stokes, so we stop after some basic circular flow examples.

Then we turn to ideal flows and Euler's equation, and spend a lot of time on incompressible, irrotational flows. Classic examples like flow past a cylinder are covered, both by solving Laplace's equation but then again in the complex plane with the Milne-Thompson circle theorem. Conformal mapping is used to handle more complex shapes, and we even discuss a little bit of aerodynamics – how does an airplane fly? Ideal line vortexes are used to model vortex pairs and vortex streets.

We end with a basic discussion of small-amplitude surface waves, including both gravity and capillary waves, as well as the classic derivation of sound waves. The last chapter in the book is devoted to special topics that I deemed too advanced or too adjacent to fit into any of the first five chapters. It's here that I actually derive the Navier-Stokes equations, talk a little bit about boundary layers, handle the classic dam breaking problem, and derive Stokes law for very viscous flow past a sphere. I even put a little bit of astrophysics in there, deriving the Lane-Emden equation and discussing polytropic models of stars. And the very last section of the very last chapter is about computational methods for fluid dynamics.

But that's just the topics; maybe you're interested in what mathematics is covered. It's a fun list: lots of vector calculus (I use the divergence theorem so many times it's burned into my brain); solving ODEs by the tried-and-true method of guessing a solution and plugging it in; solving PDEs by separation of variables (in Cartesian and cylindrical coordinates), by self-similarity, and by the method of characteristics; using the complex plane to describe basic flows, and more advanced complex analysis to interpret and advance those basic flows; linearization of nonlinear systems of equations; Fourier series and Fourier integrals to describe wave packets; and tensors in the derivation of the Navier-Stokes equation and calculation of the force on a sphere. Along the way we end up solving both a third-order ODE and a fourth-order ODE, a rarity in most physics problems.

I should mention, however, that one thing the book doesn't cover (at least beyond a couple of pages of the briefest introduction): computational methods. I'll explain why in a moment.

What else do you need to teach a course in fluid dynamics?

My feeling, after teaching this course so many times, is that fluid dynamics is really a convergence of three things:

- Theory, of which the basics is covered in this textbook.
- Coding, which is not. Although the field of computational fluid dynamics fills countless textbooks and has numerous popular software packages, I'm not even talking about *that* – I mean using code to supplement the theory. Almost every figure in the textbook required coding, and I used code to solve ODEs and do complex algebra and understand Fourier transforms. Being able to write a script that does these things is absolutely necessary for physics and math majors today. But I didn't include any code or information on how to code in the book; I wanted it to stand on its own. Instead, I created a series of Jupyter notebooks as tutorials in a number of different numerical topics. These notebooks, and the ones I created all the figures in, are available online on the web page for the book.
- Experiment, which also is not. Fluid dynamics is an incredibly rich field when it comes to experiment, and there are many things you can do easily and on your own to supplement many of the things discussed in the book. As with coding, this supplementary material can be found at the web page for the book.

Why Open Education?

This book and the supplementary material are released under a Creative Commons Attribution-ShareAlike 4.0 International License, which means you can do anything you want with this book: cut out stuff you don't need, use it to build an even better book, download it for free off the web, print it out yourself, give it to your friends (they want a copy, trust me), and anything else you can think of. The only stipulation is that you give credit where it's due, and release the derivative work under the same license.

Why did I choose this? Because I've taught university courses and have seen the issues with expensive textbooks: students not being able to afford them, students finding illegal PDF copies online and sharing them, the publishing companies fighting back with various schemes like renting digital copies, and so on. Especially in advanced physics, little textbooks can be so expensive, and yet I think textbooks are an important and necessary resource, and I want my students to all have a copy.

Thanks

Thanks to all my students over the years, but especially to Kyle Mills, who was the first (by a long shot) to write up my course notes into a book. I'm hoping this document is much improved beyond those early years of notes, but it wouldn't be here without Kyle showing me that it could be done.

Find a typo, mistake, or horrible misconception in this book? Let me know by creating a new issue at the GitHub page:

github.com/josephmacmillan/IntroductionToFluidDynamics/issues.

Chapter 1

Introduction

1.1 Fluids

What, exactly, is a fluid? Well, it's anything that *flows*, of course. But that can mean a surprising number of things.

One obvious example is water, which we'll use extensively as our go-to fluid for a number of applications. Some fluids are highly viscous – syrup is a good example – in which case the behaviour might be very different. Later on in the book we'll take an introductory look at *aerodynamics*, where the fluid is of course air.

These are all good examples of what's called a *Newtonian fluid* – the fluid has certain, fairly simple properties and can be modelled well by a set of equations called the *Navier-Stokes* equations. Other fluids, though, can be more complicated (and, for the most part, beyond the scope of this book). For example, the behaviour of fluids with a net charge (e.g., a plasma) add interesting complications that must be dealt with by including the theory of electricity and magnetism. Combining Maxwell's equations with the Navier-Stokes equations leads to the theory of magnetohydrodynamics, or MHD for short.

Even something as commonplace as blood can be beyond the Navier-Stokes equations to model. That's because blood is an example of a non-Newtonian fluid – it's nonhomogeneous and is a “shear-thinning” fluid, which means it becomes less viscous at high shear strain. Consider what happens to the blood during an anaphylactic shock – an extreme allergic reaction. The body's first response is to release histamine, which causes the blood vessels to widen. When this happens, the blood will slow down, for reasons we'll learn have to do with conservation of mass. But because blood is shear-thinning, it becomes more viscous as it goes slower. This

leads to a feedback loop – increased viscosity causes the blood to slow further, causing it to be more viscous, which means the blood slows even more, and so on. This is why anaphylaxis is so severe and needs to be treated right away with adrenaline, which increases blood flow.

As a final, possibly surprising, example, traffic can be modelled as a fluid – even though cars are large, discrete objects.

Although the study of fluid dynamics is centered around solving the Navier-Stokes equations, in this first chapter we'll begin with some preliminary ideas about the flow of fluids, in particular how we can mathematically describe and visualize it. Before we start with the heavy lifting of solving differential equations, we'll also need to be able to identify various properties of the fluid, such as its vorticity and viscosity, since in many cases understanding these can lead to significantly simpler methods of solving the equations of fluid dynamics.

1.2 Describing Fluid Flow

The main goal of fluid dynamics is to find the fluid velocity \mathbf{u} at every point \mathbf{r} at any time t :

$$\mathbf{u} = \mathbf{u}(\mathbf{r}, t). \quad (1.1)$$

This is a good time to discuss some of the notation we'll be using. First, note that I'm bolding any quantity that's a *vector*, like position and velocity. Scalars are represented by italicized letters. Secondly, I'm using \mathbf{u} to denote the velocity rather than the more usual \mathbf{v} . In fact, the letter v serves a different purpose – it's the y component of the fluid velocity. The x and z components are denoted by u and w , respectively, so we can write the full vector form of the velocity as

$$\mathbf{u} = u \hat{\mathbf{x}} + v \hat{\mathbf{y}} + w \hat{\mathbf{z}} = [u, v, w]. \quad (1.2)$$

Keep in mind that u , v , and w are each functions of x , y , z , and time:

$$u = u(x, y, z, t), \quad v = v(x, y, z, t), \quad \text{and} \quad w = w(x, y, z, t),$$

and note that I'm using some vector short hand with the square brackets.

This can sometimes be confusing notation, so be careful with it. Also note that (to add to the confusion) u does double duty: it's both the x component of the flow, as well as the name of the entire velocity vector field.

So far, I've written everything down in Cartesian coordinates, but we'll use cylindrical coordinates (s, ϕ, z) frequently as well. Figure 1.1 shows the usual cylindrical coordinate setup, and it's clear that

$$x = s \cos \phi, \quad y = s \sin \phi, \quad \text{and} \quad z = z.$$

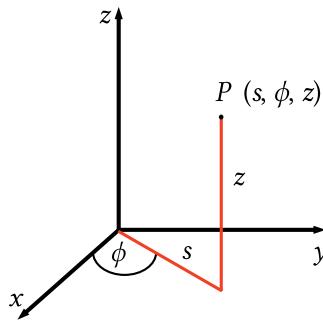


Figure 1.1: Cylindrical coordinates (s, ϕ, z) .

We can write the fluid velocity in cylindrical coordinates as

$$\mathbf{u} = u_s(s, \phi, z, t) \hat{\mathbf{s}} + u_\phi(s, \phi, z, t) \hat{\mathbf{\phi}} + u_z(s, \phi, z, t) \hat{\mathbf{z}}. \quad (1.3)$$

I'll avoid using the “bracket” shorthand if we're using cylindrical coordinates.

In much of our examination of fluid dynamics, we'll deal with special cases and symmetries which will make our job (slightly) easier. The below examples discuss two of these special cases.

Example 1.1 – Steady Flow. A *steady flow* has no explicit time dependence, so that

$$\frac{\partial \mathbf{u}}{\partial t} = 0. \quad (1.4)$$

This means that, at any point, the speed and direction of the flow are constant. We'll be dealing with this case quite frequently, especially at the beginning of the book.

Example 1.2 – Two Dimensional Flow. A *two dimensional* flow has the form

$$\mathbf{u} = [u(x, y, t), v(x, y, t), 0]. \quad (1.5)$$

Note that not only is there no z component to the velocity field, but there is furthermore no z dependence on the x and y components, either.

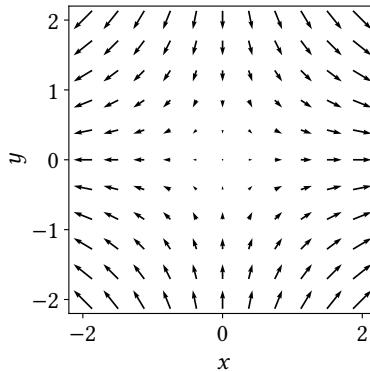


Figure 1.2: A vector plot for flow about a stagnation point. Here the parameter $\alpha = 1$. Note that at the point $(0,0)$ the flow velocity is zero; that's what makes it a stagnation point.

1.3 Visualizing Fluid Flow

1.3.1 Vector Plots

One way to visualize the flow of a fluid is by drawing a vector for \mathbf{u} at a sample of positions and, if necessary, times.

Example 1.3 – Flow about a stagnation point. For example, consider the fluid velocity vector field

$$\mathbf{u} = [\alpha x, -\alpha y, 0]. \quad (1.6)$$

This is an example of two-dimensional steady flow, and represents the flow around a *stagnation point*, which is a point in the fluid which is not flowing. The vector plot for this flow is given in Figure 1.2.

1.3.2 Streamlines

A more common way to visualize fluids is with *streamlines*. A streamline is a curve that, at time t , has the same direction as $\mathbf{u}(\mathbf{r}, t)$ at each point. This is a concept you might be familiar with from electrodynamics – electric field lines are very similar. Like field lines, streamlines can't cross, since that would imply that a particular fluid element at that point would

have two different velocities. There's one exception to this rule, of course – streamlines can cross at points in the fluid where the velocity is zero.

How do we calculate and plot streamlines? Well, we first parameterize a streamline curve, using ξ as the parameter, as

$$x(\xi), \quad y(\xi), \quad z(\xi),$$

so that, as ξ changes, the functions x , y , and z trace out the streamline curve. From the definition of a streamline, we know that

$$\frac{dx}{d\xi} = ku, \quad \frac{dy}{d\xi} = kv, \quad \frac{dz}{d\xi} = kw,$$

where k is some constant (and different streamlines will have different – but unique – values of k). Since k is the same along each streamline, we must have

$$\frac{dx/d\xi}{u} = \frac{dy/d\xi}{v} = \frac{dz/d\xi}{w}. \quad (1.7)$$

This means that, given the functional form of u , v , and w , we can find $x(\xi)$, $y(\xi)$, and $z(\xi)$.

In addition to streamlines, *pathlines* and *streaklines* are sometimes used to visualize and describe fluid flow. However, for steady flow – which we'll mostly be dealing with – they're the same as streamlines, so we won't worry about them too much. If you're interested, Problem 1.2 takes a look at an unsteady case where the path lines are different from the streamlines.

Example 1.4 – Plotting Streamlines. As an example of calculating and then plotting streamlines, let's go back to our flow about a stagnation point, Equation 1.6. Since this is two dimensional flow, we can ignore w and z dependence and look at

$$\frac{dx/d\xi}{u} = \frac{dy/d\xi}{v}.$$

Plugging in u and v for this case, and removing the parameter ξ for simplicity, gives us

$$\frac{dx}{\alpha x} = -\frac{dy}{\alpha y}.$$

Integrating both sides and solving for y gives the function

$$y(x) = \frac{c}{x}. \quad (1.8)$$

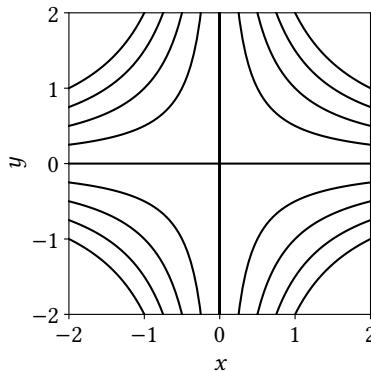


Figure 1.3: Streamlines plotted from Equation 1.8, for a variety of values of c ranging from -2 to 2 . Note that the streamlines cross at the stagnation point; that's the only place where they're allowed to.

That's our answer; c is the integration constant, which will vary from streamline to streamline. Figure 1.3 shows the streamlines plotted; compare with the vector plot in Figure 1.2.

1.4 Total Derivative and Acceleration

Now that we know how to visualize the flow of a fluid, we need to be able to calculate how it *changes*.

Let $f(x, y, z, t)$ be some quantity of interest – it might be a component of velocity (u or v , say), or maybe the density of the fluid ρ , or even the concentration c of a certain pollutant in the fluid. If we're then looking at one particular (fixed) location – say, x, y, z – in the fluid, then $\partial f / \partial t$ gives the rate of change of f at that position.

On the other hand, we might want to know how f changes as we move along with the fluid. In this case, we might imagine attaching ourselves to a certain *fluid element* – a small “blob” of the fluid – and measuring how f changes as we move along with it. This rate of change is given by

the so-called *total* or *material* derivative,¹

$$\frac{Df}{Dt} \equiv \frac{df}{dt} = \frac{d}{dt} f[x(t), y(t), z(t), t].$$

Now, $x(t)$, $y(t)$, and $z(t)$ all change as we move with the flow, and of course

$$\frac{dx}{dt} = u, \quad \frac{dy}{dt} = v, \quad \text{and} \quad \frac{dz}{dt} = w.$$

So, applying the chain rule to the derivative above, we get

$$\begin{aligned} \frac{Df}{Dt} &= \frac{\partial f}{\partial x} \frac{dx}{dt} + \frac{\partial f}{\partial y} \frac{dy}{dt} + \frac{\partial f}{\partial z} \frac{dz}{dt} + \frac{\partial f}{\partial t} \\ &= u \frac{\partial f}{\partial x} + v \frac{\partial f}{\partial y} + w \frac{\partial f}{\partial z} + \frac{\partial f}{\partial t}. \end{aligned}$$

This can be written more compactly using vectors:

$$\frac{Df}{Dt} = \frac{\partial f}{\partial t} + (\mathbf{u} \cdot \nabla) f.$$

(1.9)

Careful with that dot product; the gradient operator ∇ operates on the function f , so you can't switch the order. In this case, $(\mathbf{u} \cdot \nabla) \neq (\nabla \cdot \mathbf{u})$.

As an example of applying the total derivative, let's take f to be the fluid velocity itself:

$$\frac{D\mathbf{u}}{Dt} = \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u}.$$

(1.10)

This, as you might suspect, is the *acceleration* of the fluid element at position \mathbf{r} .

Note that if the material derivative is *zero*,

$$\frac{Df}{Dt} = 0,$$

that implies that the quantity f doesn't change *for a particular fluid element*. On the other hand, if just

$$(\mathbf{u} \cdot \nabla) f = 0,$$

¹There are many other names for this kind of derivative, too; according to Wolfram MathWorld, it's also called the convective derivative, the advective derivative, derivative following the motion, hydrodynamic derivative, and Lagrangian derivative, among a few others.

that means that f will be constant *along a streamline*. This might not be immediately obvious, so let's prove it. Let $\hat{\mathbf{e}}_s$ be a unit vector that is always parallel to a streamline in the direction of the flow. Since a streamline is always tangent to the flow, we can write

$$\mathbf{u} = |\mathbf{u}| \hat{\mathbf{e}}_s,$$

so that

$$(\mathbf{u} \cdot \nabla) f = (|\mathbf{u}| \hat{\mathbf{e}}_s \cdot \nabla) f = (|\mathbf{u}| \frac{\partial}{\partial s}) f = |\mathbf{u}| \frac{\partial f}{\partial s} = 0.$$

Thus $\partial f / \partial s = 0$ and the quantity f must be constant along the direction s , which is along the streamline.

Example 1.5 – Acceleration of the fluid. Let's return to our flow around a stagnation point. What is the acceleration of the flow?

Well, clearly $\partial \mathbf{u} / \partial t = 0$ (the flow is steady), and so

$$\frac{D\mathbf{u}}{Dt} = (\mathbf{u} \cdot \nabla) \mathbf{u} = \left(u \frac{\partial}{\partial x} + v \frac{\partial}{\partial y} \right) \mathbf{u}.$$

At this point is probably easiest to work out each vector component separately:

$$\frac{Du}{Dt} = u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \alpha^2 x,$$

and

$$\frac{Dv}{Dt} = u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = \alpha^2 y.$$

So the acceleration is

$$\frac{D\mathbf{u}}{Dt} = \alpha^2 x \hat{\mathbf{x}} + \alpha^2 y \hat{\mathbf{y}} = [\alpha^2 x, \alpha^2 y].$$

1.5 Incompressible and Irrotational Flow

1.5.1 The Incompressibility Condition

Except in one notable case – gas dynamics – we'll be dealing with fluids that are *incompressible*. This essentially just means that the density of the fluid is constant under pressure – you can't compress it. Stated another way, it means that a “dyed” blob of fluid can't change in volume as it

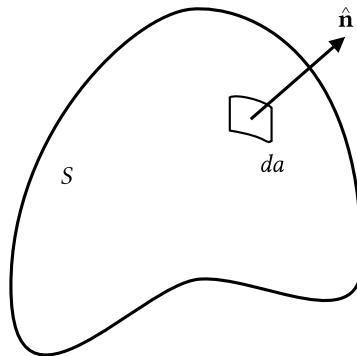


Figure 1.4: A fixed, close surface S enclosing a volume V .

moves. Water is a good example of an incompressible fluid, and we'll even treat air as incompressible in some cases.

We need to be able to write down a mathematical statement of incompressibility. To do this, consider a fixed, closed surface S in the fluid. By "fixed," I mean that it's fixed in space, so fluid can move into this closed volume V at some places and out at others; if the fluid is incompressible, though, then however much fluid is *entering* the region must equal the amount that is *leaving*.

Consider, then, an infinitesimally small piece of S – call it da (see Figure 1.4). The amount of fluid that is leaving V through da is given by

$$\mathbf{u} \cdot \hat{\mathbf{n}} da.$$

This expression, where \mathbf{u} is the fluid velocity at the location of da and $\hat{\mathbf{n}}$ points normal to the surface da in the outward direction, is a *volume flow rate*. It measures the volume of fluid that passes through da per unit time. The dot product ensures we get only that fluid which is moving *through* the tiny surface, whether it's flowing outward (positive dot product) or inward (negative dot product).

To find the total amount of fluid leaving the entire surface S , we can just integrate over that surface:

$$\int_S \mathbf{u} \cdot \hat{\mathbf{n}} da$$

is the net volume rate at which fluid is leaving. However, for an incompressible fluid, we know this must be *zero*.

To write this in a form a little more usable, we can apply the divergence theorem, which states that

$$\int_S \mathbf{F} \cdot \hat{\mathbf{n}} da = \int_V \nabla \cdot \mathbf{F} d\tau.$$

Here, \mathbf{F} is some general vector field and $d\tau$ is the infinitesimal volume element. Our incompressibility condition then becomes

$$\int_V \nabla \cdot \mathbf{u} d\tau = 0.$$

However, this must be true *anywhere* in the fluid and for any size volume; that means the integrand itself must be zero:

$$\boxed{\nabla \cdot \mathbf{u} = 0.} \quad (1.11)$$

This is the *incompressibility condition*; unless we say otherwise, every flow must obey it.

Example 1.6 – Incompressible flow. Is our flow about a stagnation point incompressible? Let's check. Recall that $\mathbf{u} = [\alpha x, -\alpha y]$. The divergence of \mathbf{u} is then

$$\nabla \cdot \mathbf{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = \alpha + (-\alpha) = 0,$$

just as we require.

1.5.2 Vorticity

The *vorticity* of a flow is defined as

$$\boxed{\boldsymbol{\omega} = \nabla \times \mathbf{u}.} \quad (1.12)$$

For a two dimensional flow $\mathbf{u} = [u(x, y), v(x, y), 0]$, we can write the vorticity as $\boldsymbol{\omega} = [0, 0, \omega]$, where

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

(note the difference between the bolded $\boldsymbol{\omega}$ and the non-bold ω – one is the full vector, and one is just the magnitude of that vector).

What, exactly, is vorticity? It's related to the *rotation* of the flow, but is a little trickier than that. A couple of examples is probably best first.

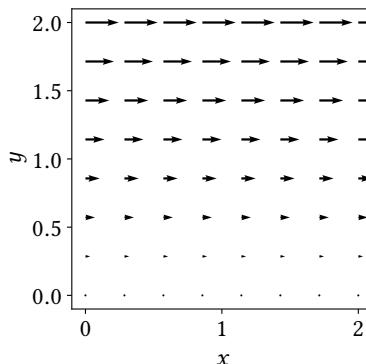


Figure 1.5: A shear flow, with $\beta = 1$.

Example 1.7 – Vorticity of shear flow. The flow given by $\mathbf{u} = [\beta y, 0, 0]$ is called a *shear flow*; a vector plot is shown in Figure 1.5. This fluid is clearly *not* rotating, but it does have vorticity. Since it's two dimensional, we can easily calculate ω :

$$\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = -\beta.$$

The vorticity is constant throughout the fluid.

Example 1.8 – Vorticity of a line vortex. Next, let's look at the flow

$$\mathbf{u} = \frac{k}{s} \hat{\phi}. \quad (1.13)$$

This is written in cylindrical coordinates, and the vector plot is shown in Figure 1.6. We'll be seeing this kind of flow quite a bit, so make sure you're familiar with it – but for now all we need to know is that the fluid is clearly rotating. Despite this, the vorticity is actually zero; I'll let you show this for homework (see Problem 1.8).

Despite these examples, vorticity really is related to the rotation of the flow – but it's the *local* rotation that it measures. If we built a simple “vorticity meter,” using two pieces of plastic and marking one tip so we can see it, the difference between the shear flow and the line vortex flow

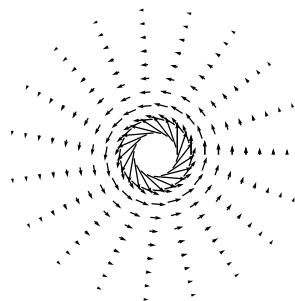


Figure 1.6: A line vortex flow, with $k = 1$.

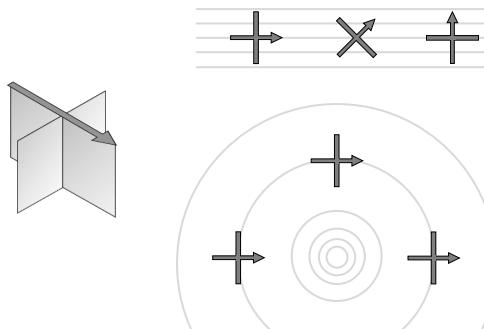


Figure 1.7: A simple vorticity meter shows the difference between two rotating flows – the shear flow (top) and line vortex flow (bottom); it measures the *local* rotation in a fluid.

is very noticeable; see Figure 1.7. In the shear flow the vorticity meter will itself rotate due to the local rotation of the fluid, while the meter in the line vortex flow will travel in a circular path but not rotate.

Finally, a bit of terminology: if the vorticity of a flow is zero, we say that it is *irrotational*, so that

$$\boxed{\nabla \times \mathbf{u} = 0} \quad (1.14)$$

for an irrotational flow.

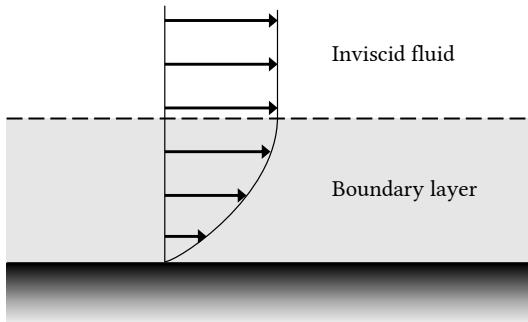


Figure 1.8: The boundary layer in a viscous fluid. The arrow here represent the fluid velocity in the horizontal direction – above the boundary layer, we can treat the fluid as inviscid, but within the velocity drops off to zero.

1.6 Viscosity

Experiment shows that a fluid doesn't "slip" along a boundary; instead, there is a thin boundary layer across which the flow speed drops smoothly and rapidly to zero (see Figure 1.8). The fluid elements in contact with the surface therefore move with the surface; this is called the *no-slip condition*.

The boundary layer comes from the viscosity of the fluid, which is defined (for so-called Newtonian fluids) via the tangential stress τ ,

$$\tau = \mu \frac{du}{dy}. \quad (1.15)$$

We'll see where this comes from later on, in Section 6.1; for now, the quantity we want to focus on is the *coefficient of viscosity* μ .

Actually, we'll more frequently use the *kinematic viscosity* ν , defined by

$$\nu \equiv \frac{\mu}{\rho},$$

where ρ is the density of the fluid. Some common viscosities (for a temperature of 15°C) are shown in Table 1.1.

In some situations, the viscosity of a fluid can be neglected; we'll call the fluid *inviscid* in that case:

$$\boxed{\nu = 0} \quad (1.16)$$

for inviscid flow. Even in situations where the viscosity is very small, though, the presence of the boundary layer can add significant complexity.

	ν (cm ² /s)
Water	0.01
Air	0.15
Olive oil	1.0
Glycerine	18
Golden syrup	1200

Table 1.1: The viscosity of some fluids.**Figure 1.9:** Boundary layer separation occurring in flow past a smooth aerofoil.
(Photo: DLR, CC-BY 3.0)

Because of the much larger velocity gradient within the boundary layer (where the fluid speed drops to zero rapidly), the viscous stress becomes significant there, even when ν is normally small enough that we could neglect viscous effects elsewhere in the flow.

There's an additional feature of boundary layers that make them important dynamically – in certain situations, they can actually separate from the boundary itself, causing the *entire* flow to be quite different than that predicted by inviscid theory. This is shown in Figure 1.9 for low-viscosity flow past an aerofoil. Inviscid theory predicts the flow to the left, but boundary layer separation causes the flow on the right, where a large wake has developed past the aerofoil.

One way to characterize how important viscosity is to the flow dynamics is with the *Reynolds number*, defined by

$$R = \frac{UL}{\nu}, \quad (1.17)$$

where U and L are velocity and length scales characteristic to the flow

(say, for example, the speed of an airplane and the width of its wing). For large Reynolds number – greater than a few thousand – viscosity can be neglected, but the flow can become unstable and *turbulent*. Below this the flow is smooth, or *laminar*, but viscosity effects can still usually be neglected. For small Reynolds number, however – less than one – viscosity becomes important in how the fluid behaves.

Problems

Problem 1.1 – Streamlines. Consider the flow given by

$$\mathbf{u} = [x, x(x - 1)(y + 1), 0].$$

Find an equation for the streamlines of the flow and plot them.

Problem 1.2 – Unsteady flow. Consider the *unsteady* flow given by

$$\mathbf{u} = [U, kt],$$

where U and k are constants. Show that the streamlines are straight lines at any particular point in time, and describe their behaviour as time goes on. Then show that the path a fluid particle follows is parabolic rather than a straight line. This is an example of how, for unsteady flow, fluid elements do not follow streamlines.

Problem 1.3 – Vector calculus practice. Calculate the divergence and curl of the following vector fields. State which flows are incompressible and which are irrotational (or both).

$$(a) \mathbf{u} = [x^2, 3xz^2, -2xz]$$

$$(b) \mathbf{u} = [xy, 2yz, 3zx]$$

$$(c) \mathbf{u} = [y^2, 2xy + z^2, 2yz].$$

Problem 1.4 – More vector calculus practice. Prove that the curl of a gradient is always zero and that the divergence of a curl is always zero.

Problem 1.5 – Concentration of a dye. Consider the flow about a stagnation point,

$$\mathbf{u} = [\alpha x, -\alpha y],$$

where α is a positive constant. Suppose a coloured dye (food colouring, for example) is introduced into the fluid, and it's concentration is given by

$$c(x, y, t) = \beta x^2 y e^{-\alpha t},$$

for $y > 0$, and where β is a constant. Does the dye concentration for any particular fluid element change with time?

Problem 1.6 – Acceleration of a rotating fluid. A fluid in uniform rotation with angular velocity Ω is given by

$$\mathbf{u} = [-\Omega y, \Omega x, 0].$$

Create a vector plot of the fluid flow, and then calculate the acceleration of the fluid and show that it can be written as

$$-\Omega^2 \mathbf{r}.$$

Is this acceleration what you expect?

Problem 1.7 – Conservation of mass. If the fluid is compressible, we can derive a condition to replace Equation 1.11 by assuming that mass must be conserved. Use arguments similar to those in Section 1.5.1 to show that it implies

$$\boxed{\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0.} \quad (1.18)$$

This is called the *continuity equation*. Show also that, if the density is constant in space and time, this reduces to the usual incompressibility condition.

Problem 1.8 – Vorticity of a line vortex. Show that the vorticity of a line vortex,

$$\mathbf{u} = \frac{k}{s} \hat{\phi},$$

is zero everywhere except at the origin, where it blows up.

Problem 1.9 – The Reynolds number. Give an order of magnitude estimate of the Reynolds number for

- (a) Flow past the wing of a Boeing 737 cruising at 150 m/s.
- (b) A thick layer of golden syrup draining off a spoon.
- (c) Air flowing through the trachea during normal breathing.
- (d) Water flowing through a creek bed.

You'll have to estimate a speed and size for most of these. Which could be turbulent? Which would require viscous flow theory, and which inviscid theory?

Chapter 2

Viscous Fluids

2.1 The Navier-Stokes Equations

We've discussed a lot of different flows already – about a stagnation point, a shear flow, a line vortex flow – but we haven't yet talked about how we *find* the fluid velocity \mathbf{u} . That is, after all, the whole goal of fluid dynamics!

In short (and we'll come back to this in Section 6.1 for a full derivation), fluid dynamics is governed by the *Navier-Stokes equation*,

$$\boxed{\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\mathbf{u} + \mathbf{g}.} \quad (2.1)$$

We can recognize some of what's there – the kinematic viscosity ν we've already talked about, and the acceleration is the left hand side. In addition, there's also:

- The *density* of the fluid, $\rho(\mathbf{r}, t)$. If the fluid is incompressible, this is constant in both space and time.
- The *pressure* of the fluid, $p(\mathbf{r}, t)$. This is a scalar function and usually not uniform throughout the fluid. Pressure is probably familiar to you, but we'll come back to it and properly define it when we derive the Navier-Stokes equations.
- The gravitational field \mathbf{g} . Usually – but not always – we'll be dealing with fluids at the surface of the Earth, and define our coordinate system so that $\mathbf{g} = [0, 0, -g]$ where $g = 9.8 \text{ m/s}^2$ as usual.

The Navier-Stokes equations, named after Claude-Louis Navier and George Gabriel Stokes who (independently) derived them, are famously

difficult to solve – they’re a set of coupled nonlinear second order partial differential equations. In fact, showing that solutions to the Navier-Stokes equation always exist and are smooth is one of the seven “Millennium Prize Problems” from the Clay Mathematics Institute. We won’t be doing anything so difficult here; for some simple situations, the equations are readily solvable, even if they require some thought and a few tricks.

2.1.1 Cartesian Coordinates

We’ll begin with flows that are best described by the usual Cartesian coordinates x , y , and z . In this coordinate system, the Navier-Stokes equations take on the form

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + g_x, \quad (2.2)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + g_y, \quad (2.3)$$

$$\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + g_z. \quad (2.4)$$

Furthermore, we’ll assume our fluids are *incompressible*, so that they must also satisfy the incompressibility condition, Equation 1.11, which in Cartesian coordinates is

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0. \quad (2.5)$$

Those are an impressively complicated looking set of equations. To solve them, we’ll first make some assumptions based on the *physical* nature of the problem we’re examining; as we’ll see in a moment, that will allow us to remove some terms and simplify things considerably.

2.2 Simple Viscous Flows

2.2.1 Poiseuille Flow in One Dimension

Consider fluid flowing steadily between two rigid boundaries, one at $y = 0$ and one at $y = h$, under a constant pressure gradient

$$\frac{dp}{dx} = -P.$$

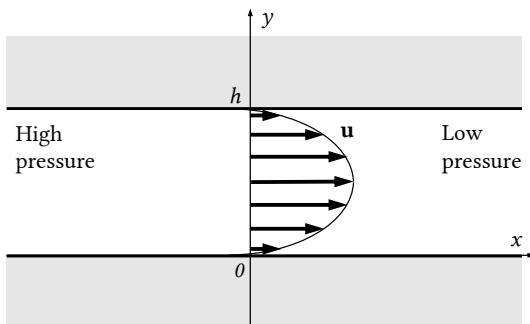


Figure 2.1: The fluid will flow to the right due to the pressure gradient.

Figure 2.1 shows the set-up and our coordinate system.

This problem was first examined by Jean Léonard Marie Poiseuille, a physiologist interested in the flow of blood though capillaries and veins,¹ although it also serves well as a model for air flowing through alveoli in lungs and fluid flowing through a straw. It's also a great introduction to solving the Navier-Stokes equations, since it's essentially a one dimensional problem, as we'll see. You can do the slightly more complicated two dimensional problem for homework (Problem 2.3).

The first step to solving the Navier-Stokes equations is to examine symmetries and dependencies. For example, this is clearly a two dimensional flow, with no dependence on z or flow in the w direction. This assumes the “walls” at $y = 0$ and $y = h$ are infinite in extent along the z -direction, but that's okay – it still makes this problem a good approximation in more realistic (finite) situations. This is also explicitly *steady* flow, so there's no time dependence at all. All this suggests we can write our the fluid velocity \mathbf{u} in terms of x and y only:

$$\mathbf{u} = [u(x, y), v(x, y), 0].$$

Furthermore, since the walls are also infinite in extent along the x -direction, we shouldn't expect any x -dependence in \mathbf{u} . If this seems surprising, consider moving our origin in Figure 2.1 left or right – since the pressure gradient is *constant*, exactly where we put the origin along the x -direction won't change the velocity at all. So that means our flow dependence is now

$$\mathbf{u} = [u(y), v(y), 0].$$

¹Actually, Poiseuille studied a slightly different problem – see Problem 2.4.

That's pretty simple, but we can do even better by looking at the incompressibility condition. In our flow, $\partial u / \partial x = 0$ since the flow only depends on y , so equation (2.5) becomes

$$\frac{\partial v}{\partial y} = 0.$$

That means that v is a constant. But remember the no-slip condition from Section 1.6 – the walls are stationary, so the fluid in contact with them can't be moving either. Since the velocity is zero at $y = 0$ and $y = h$, and v is constant throughout the fluid, $v = 0$ everywhere.

That last argument is a little tricky, but one we'll use quite a bit so go over again to make sure you get it. The result, at the end of the day, is that for this situation, our flow takes on the simple form

$$\mathbf{u} = [u(y), 0, 0]. \quad (2.6)$$

This is called steady *plane-parallel shear flow* – notice that the flow is in the x direction but the dependence is along y . Although we derived this form specific to the situation shown in Figure 2.1, it's applicable to others as well – as long as the flow is two-dimensional, steady, and has that translational symmetry along the x -direction.

Now that we know exactly what variables the flow depends on, the Navier-Stokes equations get much simpler to write out (and solve!). You should convince yourself that the x component, Equation 2.2, becomes

$$0 = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \frac{\partial^2 u}{\partial y^2}. \quad (2.7)$$

Notice there's no gravity term here; I set $g_x = 0$. In fact, for simplicity I'll assume there is no gravity at all in this problem, so $g_y = g_z = 0$, too. In Problem 2.1 you can see what effect gravity has on the situation, but it won't change the nature of it very much. The other two Navier-Stokes equations just say that the pressure doesn't depend on y or z , but we're already told that there is a constant pressure gradient in the x -direction, so that's nothing new.

Solving the differential equation in (2.7) is straightforward. Let's write $dp/dx = -P$ and rearrange to get

$$\frac{d^2 u}{dy^2} = -\frac{P}{\nu \rho}.$$

This is a second order ordinary differential equation which can be solved with two integrations to get

$$u(y) = -\frac{P}{2\nu\rho} y^2 + c_1 y + c_2,$$

where c_1 and c_2 are the two integration constants.

As usual, we can find the constants by applying our *boundary conditions*. In this case, we need the fluid at the boundary to have the same velocity as the boundary – that's the no-slip condition again. So, at the walls, we need

$$u(0) = 0 \quad \text{and} \quad u(h) = 0.$$

It follows from this that $c_2 = 0$ and

$$c_1 = \frac{Ph}{2\nu\rho},$$

and our final solution is

$$u(y) = \frac{P}{2\nu\rho}(hy - y^2). \quad (2.8)$$

The velocity profile is a parabola.

How do we visualize this flow? Streamlines won't be any good here, since the flow is entirely in the x -direction – the streamlines will just be horizontal. Instead, we can plot the parabola directly on the system as shown in Figure 2.1, where I've also drawn in some representative vectors for the velocity \mathbf{u} . In this way we can see at a glance that the flow is strongest in the middle of the walls, and drops to zero at them according to the no-slip condition.

2.2.2 Time Dependent Flow

The Poiseuille problem we just did is a great first example of solving the Navier-Stokes equations, but it's also an example of *steady flow* – nothing really changes in time. There are a few ways we can adapt this problem to introduce time dependence; we'll do two here, and you can do another one on your own if you like (Problem 2.5).

To add the time-dependence here we'll suppose that the steady flow we found in Section 2.2.1 has been going on for awhile, and at $t = 0$ the pressure gradient stops suddenly:

$$P = -\frac{dp}{dx} = 0 \quad \text{for } t > 0.$$

What happens to the fluid for $t > 0$? Well, presumably it will slowly come to rest; how long does it take, and what does the flow look like until then? To answer, we'll set up the problem the same way as above (Figure 2.1 again), with the boundaries at $y = 0$ and $y = h$. This is still a

plane parallel shear problem, but now that we have time-dependence, the Navier-Stokes equation becomes a partial difference equation,

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2}. \quad (2.9)$$

Our boundary conditions are the same as before – that $u = 0$ at the walls – but the initial condition is our steady solution, equation (2.8),

$$u(y, t=0) = \frac{P}{2\nu\rho}(hy - y^2). \quad (2.10)$$

To solve this partial differential equation, we'll use a tried and true method all physicists should know: *separation of variables*. I'll assume here you're a little bit familiar with this technique,² but I'll still go into enough detail to jog your memory.

To begin, we suppose that the solution, $u(y, t)$, can be written as the product of two single-variable functions:

$$u(y, t) = Y(y)T(t). \quad (2.11)$$

If we plug this directly into Equation 2.9 – noting that the partial derivatives treat one of the functions as a constant and act as an ordinary derivative on the other – we get

$$Y \frac{dT}{dt} = \nu T \frac{d^2 Y}{dy^2}.$$

We can rearrange this a bit to get this into a form where all the dependence on time is on the left hand side, and all the y -dependence is on the right:

$$\frac{1}{\nu T} \frac{dT}{dt} = \frac{1}{Y} \frac{d^2 Y}{dy^2}.$$

The magic of separation of variables can now happen: since t and y are independent variables – you can change one without changing the other – both sides of this equation must be constant! If that weren't the case, we could change t by a small amount, changing the left hand side, but leaving the right hand side the same – and the two sides would no longer be equal.

Let's call the constant both sides are equal to $-k^2$; we'll see why it's negative in a moment, and why having it squared is a good idea. In that

²If not, Griffiths' *Introduction to Electrodynamics* is a great place to look; he has a whole section (3.3 in the third edition) devoted to it.

case, separation of variables has turned our original partial differential equation into two ordinary differential equations:

$$\frac{dT}{dt} = -k^2 v T \quad (2.12)$$

$$\frac{d^2Y}{dy^2} = -k^2 Y. \quad (2.13)$$

The first of these equations is easy to solve; rearrange to get

$$\frac{dT}{T} = -k^2 v dt$$

and integrate. With a bit of work, we can write the solution as

$$T(t) = Ce^{-k^2 vt}, \quad (2.14)$$

where C is the integration constant. We're not quite done with this yet – we still don't know the value of C , or k for that matter – but let's move on to the second equation.

Equation (2.13) one should look familiar,³ and it has the general solution

$$Y(y) = A \sin(ky) + B \cos(ky).$$

We can apply our boundary condition to this solution right away. First, we need the fluid to be at rest along the bottom boundary at $y = 0$; for that, we'll require $Y(0) = 0$. If we evaluate the solution at $y = 0$, we get $Y(0) = B$, meaning B must be zero. Our solution therefore becomes

$$Y(y) = A \sin(ky).$$

The second boundary condition is a little trickier. Let's evaluate our solution at $y = h$:

$$Y(h) = A \sin(kh).$$

Somehow, this must equal zero as well. One possibility is to make $A = 0$ – but then $u = 0$, which doesn't make sense. Same thing with setting $k = 0$. But if we set

$$kh = n\pi, \quad n = 1, 2, 3, \dots,$$

then that will work, since the zeros of the sine function are every integer multiple of π .

³If it doesn't, time to brush up on some of your mathematics – having some familiarity with differential equations is necessary to be successful in learning fluid dynamics.

This almost completes our full solution; combining $Y(y)$ with $T(t)$ to make the velocity, we get

$$u(y, t) = Y(y)T(t) = A \sin(n\pi y/h) e^{-n^2\pi^2 vt/h^2},$$

where I've grouped both constants into A . But wait! This is actually an *infinite* number of solutions, since n is any integer. In fact, we can write down the *general* solution to our original partial differential equation as a superposition of all of these individual solutions,

$$u(y, t) = \sum_{n=1}^{\infty} A_n \sin(n\pi y/h) e^{-n^2\pi^2 vt/h^2}. \quad (2.15)$$

Note that the constant A could be different in each term in the sum, so it has an index n on it.

All we have left to do is find the values of the A_n s, which we do by using the initial condition. In fact, our solution can fit *any* initial condition we want; that's part of the magic of separation of variables. If we evaluate equation (2.15) at $t = 0$ and equate it to our initial conditions, equation (2.10), we get

$$\sum_{n=1}^{\infty} A_n \sin(n\pi y/h) = \frac{P}{2v\rho} (hy - y^2).$$

Somehow we have to solve this equation for A_n . We can do that by application of what is sometimes called *Fourier's trick*:⁴ we'll multiply both sides by $\sin(m\pi y/h)$ and integrate from $y = 0$ to $y = h$, giving

$$\sum_{n=1}^{\infty} A_n \int_0^h \sin(m\pi y/h) \sin(n\pi y/h) dy = \frac{P}{2v\rho} \int_0^h (hy - y^2) \sin(m\pi y/h) dy. \quad (2.16)$$

Now, the left hand side turns out to be zero if $n \neq m$; the two sine functions are called *orthogonal* in that case. In the case where $n = m$, the integral reduces to

$$\int_0^h \sin^2(m\pi y/h) dy = \frac{h}{2}.$$

We can combine these two results into one, using the Kronecker delta,

$$\delta_{mn} = \begin{cases} 0 & \text{if } m \neq n, \\ 1 & \text{if } m = n. \end{cases} \quad (2.17)$$

⁴In fact, you might recognize these techniques as a part of the general theory of Fourier series.

Thus the left hand side of equation (2.16) becomes

$$\sum_{n=1}^{\infty} A_n \int_0^h \sin(m\pi y/h) \sin(n\pi y/h) dy = \frac{h}{2} \sum_{n=1}^{\infty} A_n \delta_{mn}.$$

But the Kronecker delta will *collapse* the sum – every term in it will be zero except the m th one, so we get

$$\frac{h}{2} \sum_{n=1}^{\infty} A_n \delta_{mn} = \frac{h}{2} A_m.$$

Pew! That takes care of the left hand side, but we still have to do the right side. That's a bit messy, since

$$\int_0^h (hy - y^2) \sin(m\pi y/h) dy = \frac{2h^3}{\pi^3 m^3} (1 - \cos(m\pi)).$$

Why messy? Because the cosine evaluates to either +1 (if m is even, and we get zero) or -1 (if m is odd, and we get 2). That means we have to handle both of those cases separately. Putting both sides of equation (2.16) together and solving for A_m gives

$$A_m = \begin{cases} 0, & m \text{ even}, \\ 4Ph^2/\pi^3 \nu \rho m^3, & m \text{ odd}. \end{cases} \quad (2.18)$$

Finally, we can write down our full solution to this problem. Inserting this expression for A_m into equation (2.15) gives us

$$u(y, t) = \frac{4Ph^2}{\pi^3 \nu \rho} \sum_{n=1,3,5,\dots}^{\infty} \frac{1}{n^3} \sin(n\pi y/h) e^{-n^2 \pi^2 v t / h^2}. \quad (2.19)$$

That's not particularly nice looking answer, and it took us some work to get there, but we're done. This gives us the fluid velocity at any time t between the walls.

To take a look at the velocity, let's plot it up at a few times t , using the same convention we used in Figure 2.1, where the distance along the y -direction is on the vertical axis, and the fluid velocity is along the horizontal. Figure 2.2 shows the flow at three times, one of which is $t = 0$. It's clear that, over time, the velocity does in fact go to zero. If we want to estimate the length of time it takes to come to rest, we can look just at the $n = 1$ term in the solution – it's the dominant term, since each term

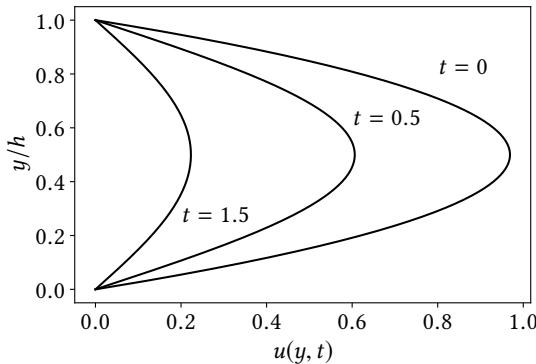


Figure 2.2: Poiseuille flow is slowing down over time once the pressure gradient shuts off. The velocity is in units of $4Ph^2/\pi^3v\rho$, and the time is in units of the characteristic time $\tau = h^2/\pi^2v$. This plot used about 15 terms in the sum in Equation 2.19.

after it gets smaller by a factor of $1/n^3$. In that case, we can write the time factor as

$$e^{-\pi^2vt/h^2} = e^{-t/\tau},$$

where $\tau = h^2/\pi^2v$ is the characteristic time. Thus the speed in the channel slows to about 37% of its initial amount in a time τ . By a few characteristic times the fluid is practically at rest.

2.2.3 An Impulsively Moved Boundary

I'd like to do one more example with the two rigid boundaries before moving on – mostly because in this example there's an interesting twist with how we'll have to solve the partial differential equation.

This time, we'll start with the fluid at rest in between the two boundaries; the set up is the same as in the above examples and once again Figure 2.1 shows the coordinate system. In this case, though, there won't be any pressure gradient – we'll take $dp/dx = 0$ for the whole problem. Once again, since nothing changes physically about the problem, we have a fluid dependent only on y and flowing only in the x -direction, so this is still plane-parallel shear flow.

We'll put the fluid in motion by, at $t = 0$, suddenly *jerking* the bottom boundary (the one at $y = 0$) – we'll pull it to the right with constant speed U . There won't be a period of acceleration here – it's *impulsively* moved and goes from at rest to moving instantaneously. Thanks to the no-slip theorem, the bottom boundary will pull on the fluid elements in

contact with it, which will in turn exert viscous forces on the fluid above, and so on. To find out how the fluid responds to the impulsively moved boundary, we'll need to once again solve the Navier-Stokes equation.

It's identical, though, to the above time-dependent problem, equation (2.9),

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2}. \quad (2.20)$$

The initial condition, however, is that the fluid starts at rest,

$$u(y, 0) = 0. \quad (2.21)$$

The boundary conditions come from the no-slip condition, with the top boundary at rest but the bottom moving at constant velocity:

$$u(0, t) = U \quad \text{for } t > 0, \quad (2.22)$$

$$u(h, t) = 0. \quad (2.23)$$

To solve the differential equation (2.20), we'll once again use technique of separation of variables. However, there's a problem in this case – the boundary condition is nonhomogeneous (since $u(0, t)$ is non-zero). Unfortunately, separation of variables breaks down with nonhomogeneous boundary conditions (go ahead and try it on your own!).

We can fix up the problem to deal with that, though, using a standard technique. First, let's imagine that the flow is actually *steady* – there's no time dependence. In that case, our equation simply becomes

$$\frac{d^2 u_{\text{steady}}}{dy^2} = 0,$$

which has the solution

$$u_{\text{steady}}(y) = c_1 + c_2 y.$$

To fit the boundary conditions, we need $c_1 = U$ and $c_2 = -U/h$, so we have

$$u_{\text{steady}}(y) = U \left(1 - \frac{y}{h}\right).$$

This flow satisfies the differential equation and the boundary conditions, but not the initial conditions – there's no time dependence at all. However, it might be reasonable to think that, after a long time has passed, the flow will eventually reach a steady state like this. In that case, it makes sense to write the *full* solution as

$$u(y, t) = u_{\text{steady}}(y) + u_1(y, t),$$

where $u_1(y, t)$ is an unknown “transient” function; clearly, we want $u_1 \rightarrow 0$ as $t \rightarrow \infty$ so that we get our long-time steady state.

If we plug this form of $u(y, t)$ into Equation 2.20, our differential equation becomes

$$\frac{\partial u_1}{\partial t} = v \frac{\partial^2 u_1}{\partial y^2}. \quad (2.24)$$

Notice that u_{steady} has disappeared from the differential equation, and it looks like we’ve just swapped out u for u_1 . What have gained? Well, u_1 has different boundary conditions than u – notably, it now goes to zero at both $y = h$ and $y = 0$, where the steady state solution cancels the original condition – and our boundary conditions are now homogeneous:

$$u_1(0, t) = 0 \quad \text{for } t > 0, \quad (2.25)$$

$$u_1(h, t) = 0. \quad (2.26)$$

The initial condition has also changed, since we need to cancel off the steady state term there, too:

$$u_1(y, 0) = -u_{\text{steady}} = -U \left(1 - \frac{y}{h}\right). \quad (2.27)$$

Now we’re ready to solve by separation of variables. The process and most of the steps are identical to what we just went over in Section 2.2.2, so I’ll skip some of the details. Thanks to the same differential equation and boundary conditions for u_1 , the solution is once again

$$u_1(y, t) = \sum_{n=1}^{\infty} A_n \sin(n\pi y/h) e^{-n^2\pi^2 vt/h^2}.$$

All that’s left is to find the A_n s that fit the initial conditions, equation (2.27). Using Fourier’s trick again (multiplying by $\sin(m\pi y/h)$ and integrating) leads to

$$A_n = -\frac{2U}{n\pi}.$$

Finally, adding in the steady-state solution to get $u(y, t)$, we have

$$u(y, t) = U \left[1 - \frac{y}{h} - \frac{2}{\pi} \sum_{n=1}^{\infty} \frac{1}{n} \sin(n\pi y/h) e^{-n^2\pi^2 vt/h^2} \right].$$

Figure 2.3 shows the fluid velocity $u(y, t)$ at a couple of different times. The characteristic time is the same as in the previous problem; after a few $\tau = h^2/\pi^2 v$, the fluid reaches its steady state.

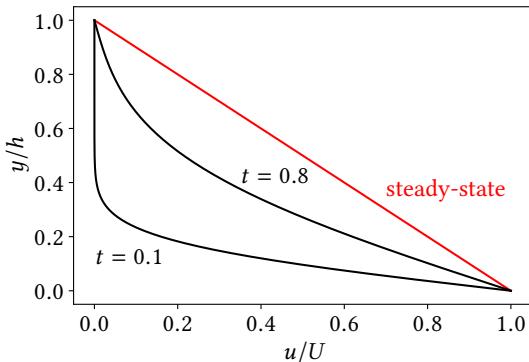


Figure 2.3: A boundary suddenly jerked into motion will eventually move the fluid into a steady state. The time is in units of the characteristic time τ .

2.2.4 Self-Similar Flows

The techniques we used to solve the three previous problems – integrating an ordinary differential equation, using separation of variables to solve a partial differential equation, and then doing the same thing but with the added twist of using a steady state solution to handle nonhomogeneous boundary conditions – are all standard techniques physicists use to solve differential equations. All three problems also had very similar set-ups – two boundaries and plane parallel shear flow.

I'd like to do one last example of time-dependent plane parallel shear flow before moving on to other kinds of situations, but this time I'll remove the boundary at $y = h$ – we only have one boundary in this new problem, and it's along the x -axis. Fluid exists only in the region $y > 0$, but it can extend up to infinity – see Figure 2.4.

As with the previous example, this boundary will be impulsively put into motion at $t = 0$ with a velocity (to the right) of U . We've already gone through the arguments in the last section about what kind of flow this is, and they all hold here, too – the only difference is in the boundary conditions. That means we once again have the partial differential equation

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2} \quad (2.28)$$

to solve for our system. The initial condition is just that the fluid is at rest,

$$u(y, 0) = 0, \quad (2.29)$$

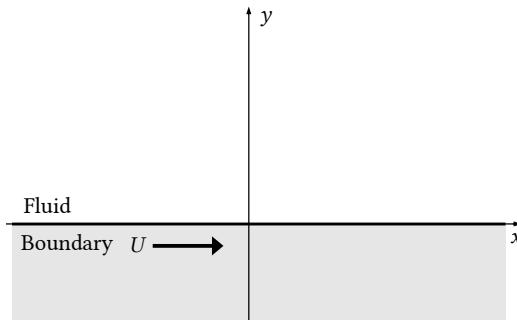


Figure 2.4: A single boundary is suddenly jerked into motion. This problem is *self-similar* because it doesn't have a scale.

and our usual no-slip condition at the boundary still applies,

$$u(0, t) = U. \quad (2.30)$$

We have one other condition, as well, for the fluid far away from the boundary; this is where our problem will start to deviate from the others. We'd expect that the motion of the boundary will affect the fluid above it, and this motion will propagate upward over time. However, the fluid *infinitely* far away will need an infinite amount of time to feel the motion of the boundary, which suggests that we require

$$u(\infty, t) = 0. \quad (2.31)$$

Now, we could, if we wanted, go through the same procedure as before – find a steady-state solution, apply separation of variables, and build our final answer as a superposition of solutions. But that process is cumbersome (and we've already done it once!), and this problem can actually be solved in a completely different way, first done by Stokes.⁵

Consider, first, some dimensional analysis. Examining equations (2.28)-(2.31) we can see that our problem depends, at best, on four variables: U , ν , y , and t . But if we write the velocity as a non-dimensional variable,

$$\tilde{u} = \frac{u}{U},$$

then our differential equation becomes

$$\frac{\partial \tilde{u}}{\partial t} = \nu \frac{\partial^2 \tilde{u}}{\partial y^2}. \quad (2.32)$$

⁵This example is actually known as Stokes' first problem.

More importantly, our boundary condition at $y = 0$ becomes

$$\tilde{u}(0, t) = 1.$$

This means that the solution for the non-dimensional \tilde{u} can only depend on *three* variables: v , y , and t . I'll write out the solution to the Navier-Stokes equation, then, as

$$\tilde{u} = f(v, y, t),$$

where f is some unknown function. We do know one thing about it, though: since \tilde{u} is dimensionless, so too must be f .

Okay, where are we going with this? Well, since f must be dimensionless, and since it can only depend on v , y , and t and no other quantity, we need to rearrange those three variables into a combination that is dimensionless. It turns out there is only *one* way to do this, the combination

$$\frac{y}{\sqrt{vt}}.$$

All this is to suggest that, in our final solution to equation (2.32), y and t and v can *only* appear in the above combination. This powerful idea will allow us to turn the partial differential equation into an ordinary one that we can integrate to solve – much simpler than going through separation of variables.

This whole idea – having the solution depend only on a dimensionless quantity – is called *self-similarity*, for reasons we'll see in a bit. It's due to the fact that there is no *physical scale* involved in the problem – it doesn't involve a length scale at all, nor a time scale. This is in contrast to the previous problem with a boundary at $y = h$ – the value of h sets the physical scale along the y -direction, and we wouldn't be able to write down a single combination of variables that was dimensionless in that case. If you're finding it difficult to imagine the scale-free nature of the problem, imagine “zooming” in or out in Figure 2.4 – because the fluid goes to infinity above the boundary, there's nothing to tell you that you actually *have* zoomed in or out, and the figure would look exactly the same.

Self-similarity is a powerful concept in fluid dynamics and many other fields, from mathematics to finance. The most common example of it is probably the nature of some fractals, such as the Koch snowflake shown in Figure 2.5. As you zoom in on the snowflake, the perimeter continues to look exactly the same. A great example of a self-similar system in nature is the fronds on a fern, as shown in Figure 2.6. Note that the overall structure of the fern is repeated on smaller scales, all the way down to

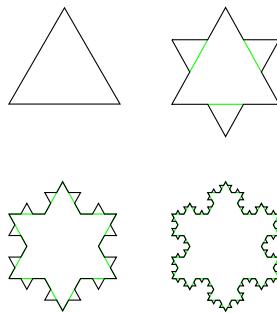


Figure 2.5: The procedure for constructing a Koch snowflake, a good example of exact self-similarity. (Image: Wxs, CC BY-SA 3.0.)

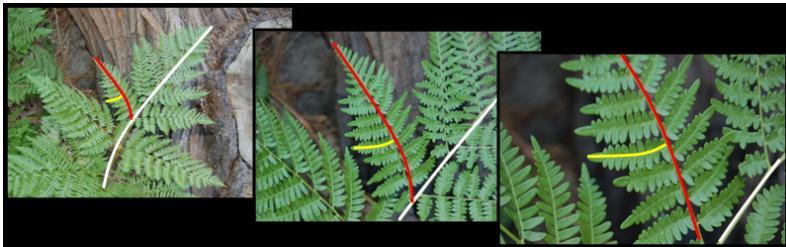


Figure 2.6: A fern is an example of self-similarity found in nature. (Photo: Jason Hine, CC BY-NC 2.0.)

the individual leaves. As we'll see, along with the scale-free situation, our solution to this problem will be self-similar in the same way.

It's time to exploit this idea mathematically. Let's define a new variable,

$$\eta = \frac{y}{\sqrt{vt}}. \quad (2.33)$$

Our goal will be to rewrite the partial differential equation (2.32) in terms of η rather than y and t . We can do this with the chain rule, so that

$$\frac{\partial \tilde{u}}{\partial t} = \frac{df}{d\eta} \frac{\partial \eta}{\partial t} = -\frac{y}{2v^{1/2}t^{3/2}} f',$$

where the prime indicates a derivative with respect to η , and

$$\frac{\partial^2 \tilde{u}}{\partial y^2} = \frac{d^2 f}{d\eta^2} \left(\frac{\partial \eta}{\partial y} \right)^2 + \frac{\partial f}{\partial \eta} \frac{\partial^2 \eta}{\partial y^2} = \left(\frac{1}{vt} \right) f''.$$

Careful with the second derivative calculation – it uses a product rule as well as the chain rule, so go over it yourself to make sure you know where the terms come from. Plugging these two derivatives back into equation (2.32) gives, after a bit of rearranging,

$$f'' + \frac{1}{2}\eta f' = 0. \quad (2.34)$$

As promised, we now have only one ordinary differential equation to worry about – and note that this equation contains only dimensionless quantities.

It's straightforward to solve this differential equation as well. First, let

$$g = f',$$

so that $g' = f''$, and the equation becomes the first order

$$g' + \frac{1}{2}\eta g = 0.$$

Writing all the gs on one side and the ηs on the other allows us to integrate it, to get

$$g(\eta) = f' = Be^{-\frac{1}{4}\eta^2},$$

where B is the integration constant. Integrating one more time will get us $f(\eta)$,

$$f(\eta) = A + B \int_0^\eta e^{-\frac{1}{4}\eta'^2} d\eta'.$$

Here, A is another integration constant (to account for the definite integral), and I've written η' as the integration variable to avoid confusion. Unfortunately, actually solving the integral in this equation is not possible in closed form – in fact, the integral

$$\frac{1}{\sqrt{\pi}} \int_0^x e^{-\frac{1}{4}u^2} du \equiv \text{erf}(x),$$

is called the *error function*, and is used extensively in probability and statistics. Since we can only evaluate the error function numerically, I'll continue to just write out the integral.

We can find the constants A and B with our boundary conditions and the initial condition. Before we do that, though, we need to express them in terms of η rather than y and t . The first boundary condition is

$$\tilde{u}(0, t) = 1.$$

But at this place ($y = 0$) and time (t), we have $\eta = 0$, so the boundary condition for $\eta = 0$ becomes

$$f(0) = 1.$$

The second boundary condition likewise becomes

$$f(\infty) = 0.$$

Also, recall that the initial condition was $\tilde{u}(y, 0) = 0$. In this case, $\eta \rightarrow \infty$, since t is in the denominator, and we get again the same expression as the second boundary condition – as expected, we have only two boundary conditions for our second order differential equation.

Using these two boundary conditions leads to $A = 1$ and $B = -1/\sqrt{\pi}$. We can write our final solution, then (using $u = U\tilde{u}$), as

$$u(y, t) = f(\eta) = U \left[1 - \frac{1}{\sqrt{\pi}} \int_0^{\eta} e^{-\frac{1}{4}\eta'^2} d\eta' \right], \quad (2.35)$$

where, of course, $\eta = y/\sqrt{vt}$.

We can take a look at the velocity for a couple of different times to see how the fluid responds to the impulsively moved boundary – see Figure 2.7. As expected, at later time the fluid is moving faster, and *more* of the fluid is moving. This is evident in Figure 2.7; at $t = 1$, the fluid above $y \sim 3$ practically isn't moving, while for $t = 10$, the motion of the fluid extends up to $y \sim 10$.

I mentioned earlier that this solution would be self-similar, but that's not very noticeable in Figure 2.7. That's because both time *and* space must be scaled together to get the same velocity curve. In fact, if we zoom out on the y -axis as we move forward in time, the motion of the fluid should always be the same. We can see that in Figure 2.8, which changes the scale of the y -axis for the right-hand plot at $t = 10$. Rather than going from $y = 0$ to $y = 10$, as in Figure 2.7, we go from $y = 0$ to

$$y = 10\sqrt{vt}$$

(remember, $\eta = y/\sqrt{vt}$). Thus, as t changes, so too does the scale of y . In this case, both velocity curves look identical, and the solution is self-similar.

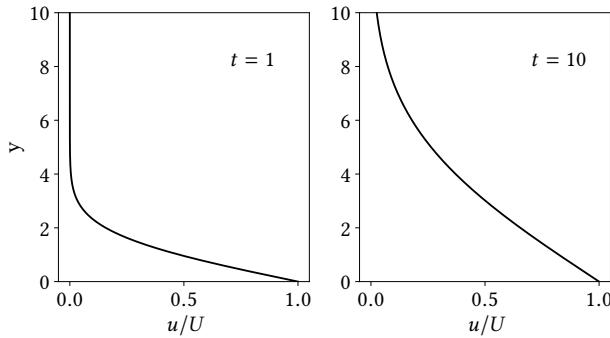


Figure 2.7: The fluid velocity for an impulsively moved boundary at $y = 0$. Note how long it takes for the fluid to respond – at the later time on the right (ten times longer than the left), the velocity has increased at all points but still drops to zero around $y = 10$. I set $\nu = 1$ for convenience.

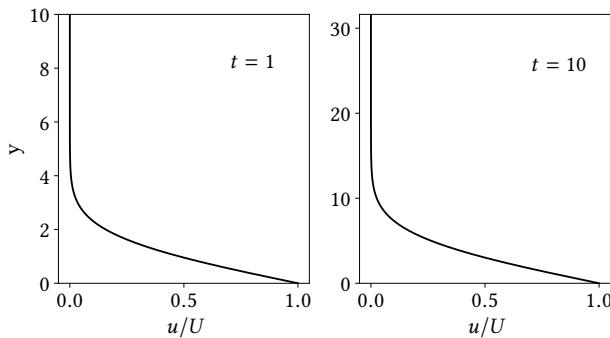


Figure 2.8: The fluid velocity for an impulsively moved boundary at $y = 0$. This time, I've changed the y -axis scale so that the two curves are self-similar.

2.3 Circular Flow

2.3.1 The Navier-Stokes Equations in Cylindrical Coordinates

The Navier-Stokes equations are trickier to write down in cylindrical coordinates, mainly due to the fact that the cylindrical unit vectors \hat{s} and $\hat{\phi}$ in equation (1.3) are not *constant* – they depend on ϕ :

$$\begin{aligned}\hat{r} &= \cos \phi \hat{x} + \sin \phi \hat{y}, \\ \hat{\phi} &= -\sin \phi \hat{x} + \cos \phi \hat{y}.\end{aligned}$$

When evaluating the derivatives in the Navier-Stokes equation, then, you have to be very careful to take this dependence into account. For example, note that

$$\frac{d\hat{s}}{d\phi} = \hat{\phi}$$

and

$$\frac{d\hat{\phi}}{d\phi} = -\hat{s}.$$

Furthermore, the gradient and Laplacian operators in cylindrical coordinates are a little more cumbersome than in Cartesian; I won't bother writing them out here, but you should look them up on your own.⁶ Altogether, here's the Navier-Stokes equations in cylindrical coordinates for you:

$$\frac{\partial u_s}{\partial t} + (\mathbf{u} \cdot \nabla) u_s - \frac{u_\phi^2}{s} = -\frac{1}{\rho} \frac{\partial p}{\partial s} + \nu \left(\nabla^2 u_s - \frac{u_s}{s^2} - \frac{2}{s^2} \frac{\partial u_\phi}{\partial \phi} \right) + g_s \quad (2.36)$$

$$\frac{\partial u_\phi}{\partial t} + (\mathbf{u} \cdot \nabla) u_\phi + \frac{u_s u_\phi}{s} = -\frac{1}{\rho s} \frac{\partial p}{\partial \phi} + \nu \left(\nabla^2 u_\phi - \frac{u_\phi}{s^2} + \frac{2}{s^2} \frac{\partial u_s}{\partial \phi} \right) + g_\phi \quad (2.37)$$

$$\frac{\partial u_z}{\partial t} + (\mathbf{u} \cdot \nabla) u_z = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \nabla^2 u_z + g_z \quad (2.38)$$

$$(2.39)$$

where

$$(\mathbf{u} \cdot \nabla) = u_s \frac{\partial}{\partial s} + \frac{u_\phi}{s} \frac{\partial}{\partial \phi} + u_z \frac{\partial}{\partial z}, \quad (2.40)$$

$$\nabla^2 = \frac{1}{s} \frac{\partial}{\partial s} \left(s \frac{\partial}{\partial s} \right) + \frac{1}{s^2} \frac{\partial^2}{\partial \phi^2} + \frac{\partial^2}{\partial z^2}. \quad (2.41)$$

⁶I really like the front and back covers in Griffiths' *Introduction to Electrodynamics* for this.

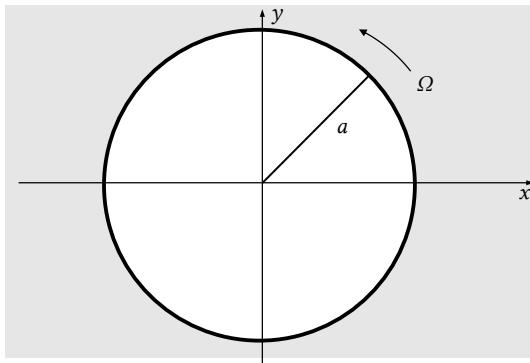


Figure 2.9: The fluid is constrained to be within a cylinder of radius a . We'll put the origin at the centre of the cylinder.

In addition, the incompressibility condition $\nabla \cdot \mathbf{u} = 0$ becomes

$$\frac{1}{s} \frac{\partial}{\partial s} (su_s) + \frac{1}{s} \frac{\partial u_\phi}{\partial \phi} + \frac{\partial u_z}{\partial z} = 0. \quad (2.42)$$

2.3.2 Uniformly Rotating Fluid

For our first problem, we'll consider fluid within a cylinder of radius a , as shown in Figure 2.9, with the cylinder rotating with an angular speed Ω .

Already this is two dimensional flow – we'll assume the cylinder is infinitely long, so there won't be any z dependence of the flow in that direction. To make things simpler, we'll also impose full cylindrical symmetry on our flow: the velocity will always take the form

$$\mathbf{u} = u_\phi(s, t) \hat{\phi}. \quad (2.43)$$

This means that the flow is always *circular*, with streamlines that are circles. Furthermore, the speed of the fluid only depends on the distance s from the origin, giving us the cylindrical symmetry. One nice benefit to a flow of this form is that the incompressibility condition is automatically satisfied, which you can check yourself with Problem 2.7.

As you might expect, taking this form of the velocity drastically re-

duces the Navier-Stokes equations. Equations 2.36 - 2.38 become

$$\frac{u_\phi^2}{s} = \frac{1}{\rho} \frac{\partial p}{\partial s} \quad (2.44)$$

$$\frac{\partial u_\phi}{\partial t} = -\frac{1}{\rho s} \frac{\partial p}{\partial \phi} + \nu \left(\frac{\partial^2 u_\phi}{\partial s^2} + \frac{1}{s} \frac{\partial u_\phi}{\partial s} - \frac{u_\phi}{s^2} \right) \quad (2.45)$$

$$0 = -\frac{1}{\rho} \frac{\partial p}{\partial z}. \quad (2.46)$$

Note that I've completely dropped the gravity terms here – but see Problem 2.1 for the minimal effect that gravity has on our problems.

Now, the third equation above – the one for z – just says that the pressure p doesn't depend on z at all, not surprising given the symmetry of the problem. The first equation gives us a relationship between the fluid velocity u_ϕ and the radial pressure gradient, so if we know one we can get the other. The second equation is the important one, but we can clean it up a bit more.

Since $u_\phi = u_\phi(s, t)$ only, equation (2.45) says that $\partial p / \partial \phi$ must also be a function of only s and t . So we can write

$$\frac{\partial p}{\partial \phi} = P(s, t),$$

where $P(s, t)$ is some unknown function. If we integrate this to find p , we get

$$p = P(s, t)\phi + f(s, t),$$

where $f(s, t)$ is the integration “constant” – remember, we have a partial derivative here (and note that there's no z dependence thanks to equation (2.46)). But this shows us that the function $P(s, t)$ *must* be zero, since otherwise the pressure would be a multivalued function of position; it would take on different values for $\phi = 0$ and $\phi = 2\pi$, for example, even though those two angles represent the same point in space.

Equation (2.45) then becomes, with $P(s, t) = \partial p / \partial \phi = 0$,

$$\boxed{\frac{\partial u_\phi}{\partial t} = \nu \left(\frac{\partial^2 u_\phi}{\partial s^2} + \frac{1}{s} \frac{\partial u_\phi}{\partial s} - \frac{u_\phi}{s^2} \right)}. \quad (2.47)$$

This equation determines the evolution of the fluid velocity, and holds for any flow with circular streamlines.

Now we can go back to our original problem – fluid inside a rotating cylinder. We'll first assume the fluid is *steady*, so that $\partial u_\phi / \partial t = 0$.

Multiplying equation (2.47) by s^2 gives us

$$s^2 \frac{d^2 u_\phi}{ds^2} + s \frac{du_\phi}{ds} - u_\phi = 0. \quad (2.48)$$

Notice that the viscosity has dropped out of the problem. To solve this ordinary differential equation, let's "guess" at a power-law solution of the form

$$u_\phi(s) = s^m.$$

Plugging this form into our equation leads to the quadratic

$$m^2 - 1 = 0,$$

which has roots $m = +1$ and $m = -1$. Our general solution is then

$$u_\phi(s) = As + \frac{B}{s}. \quad (2.49)$$

Since we have viscous fluid, one of our boundary conditions must be that the fluid moves with the same velocity as the cylinder wall, so that

$$u_\phi(a) = \Omega a.$$

We don't really have another boundary condition, since there isn't another boundary; however, note that the second term in Equation 2.49 blows up at $s = 0$, the centre of the cylinder. So, to keep the fluid velocity everywhere finite, we need to set $B = 0$. The boundary condition then leads to $A = \Omega$, so our solution is

$$u_\phi(s) = \Omega s. \quad (2.50)$$

This flow is in *solid body rotation* – in other words, the fluid rotates in exactly the same way as a solid body would, at constant angular speed Ω .

2.3.3 Spin Down of Rotating Fluid

What happens to the fluid if the rotating cylinder is suddenly brought to rest? Now we have time-dependent flow, and we'll have to use the full equation (2.47) to model what happens. Just a warning before we proceed: this is a pretty complicated problem to solve, and will require some knowledge about Bessel functions. We'll take it slow, but it might help to read up on them first.

We'll use separation of variables to solve the partial differential equation, but first we should write out the initial and boundary conditions.

The initial condition is of course our steady state solution we just found, equation (2.50),

$$u_\phi(s, t = 0) = \Omega s. \quad (2.51)$$

We'll suppose the cylinder is brought to rest at $t = 0$, so after that the no-slip condition says that

$$u_\phi(a, t) = 0 \quad \text{for } t > 0. \quad (2.52)$$

As with our last problem, we'll also have to make sure that the velocity is finite everywhere in the fluid.

Let

$$u_\phi(s, t) = S(s)T(t)$$

and plug that into equation (2.47). Dividing both sides by ST gives

$$\frac{1}{vT} \frac{dT}{dt} = \frac{1}{S} \frac{d^2S}{ds^2} + \frac{1}{sS} \frac{dS}{ds} - \frac{1}{s^2}.$$

The left hand side is a function of time t only, while the right hand side depends only on s . Since they're independent variables, both sides must be equal to a constant, which, with the benefit of hindsight, we'll call $-k^2$.

The left hand side becomes

$$\frac{dT}{dt} = -vk^2T,$$

which has the solution

$$T(t) = Ce^{-vk^2t}, \quad (2.53)$$

where C is a constant. The right hand side is a little more difficult. With a little manipulation (multiplying both sides by s^2S) we can write it as

$$s^2 \frac{d^2S}{ds^2} + s \frac{dS}{ds} + (k^2 s^2 - 1) S = 0.$$

One more small change: let

$$x = ks.$$

Then the equation becomes

$$x^2 \frac{d^2S}{dx^2} + x \frac{dS}{dx} + (x^2 - 1) S = 0. \quad (2.54)$$

This equation isn't easy to solve by hand, but it turns out to be a fairly famous differential equation that pops up in various places in physics (like electromagnetism and acoustics) – it's called Bessel's equation, and the solutions are called *Bessel functions*.

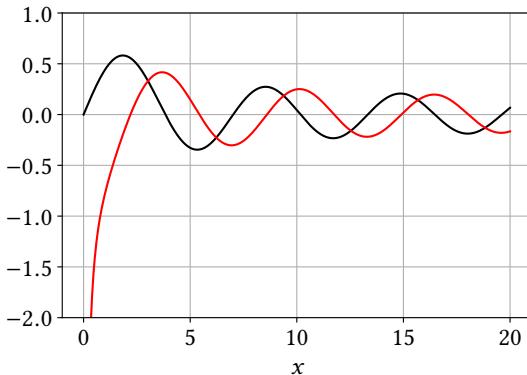


Figure 2.10: The Bessel functions of the first ($J_1(x)$, black) and second ($Y_1(x)$, red) kind, of order $\alpha = 1$.

	$J_1(x)$	$Y_1(x)$
λ_1	3.83170597	2.19714133
λ_2	7.01558667	5.42968104
λ_3	10.17346814	8.59600587
λ_4	13.32369194	11.74915483
λ_5	16.47063005	14.89744213

Table 2.1: The first five zeros for the Bessel functions.

The Bessel function of the first kind is usually written $J_\alpha(x)$, where α is called the *order* of the Bessel function. In our case, with the $(x^2 - 1)$ term in the equation, we're dealing only with the first order, so $\alpha = 1$. The second solution is the Bessel function of the second kind, sometimes called the Neumann function, $Y_\alpha(x)$. We can't write either function in closed form – the best we can do is a series solution – but I've plotted both $J_1(x)$ and $Y_1(x)$ in Figure 2.10. Note that both functions are oscillatory, but not with a nice regular period like sine and cosine; Table 2.1 lists a few of the zeros of each, and we'll need those in a moment.

The general solution to Equation 2.54 is the superposition of both Bessel functions,

$$S(x) = AJ_1(x) + BY_1(x).$$

However, we can see from Figure 2.10 that $Y_1(x)$ goes to negative infinity at the origin. This is no good – we want the velocity of the fluid to always be finite – so we can drop the this function and set $B = 0$. Going back to

using s instead of x , we have

$$S(s) = AJ_1(ks).$$

Now we can apply our boundary condition at $s = a$. I'll write it out for you:

$$S(a) = AJ_1(ka) = 0.$$

Clearly, we can't set $A = 0$, since that will give us a static fluid; instead, we'll have to use the fact that the Bessel function has an infinite number of zeros, just like we did with the sine function back in Section 2.2.3. Unfortunately, the Bessel zeros aren't at nicely periodic locations like $n\pi$; as you can see in Table 2.1, they're neither periodic nor at nice locations. Instead, we'll have to be more general, so I'll call the n th zero of the Bessel function λ_n . That is,

$$J_1(\lambda_n) = 0$$

for every value of λ_n . The boundary condition thus implies that $ka = \lambda_n$, or

$$k = \frac{\lambda_n}{a}.$$

Let's put our solution so far together. We'll combine the results for $T(t)$ and $S(s)$ to get the fluid velocity

$$u_\phi(s, t) = S(s)T(t) = A J_1(\lambda_n s/a) e^{-\lambda_n^2 vt/a^2},$$

where I've combined the two constants into A . As is usual with separation of variables, we end up with not just *one* solution, but an *infinite* number of them. That means the most general solution is a linear combination of all of them,

$$u_\phi(s, t) = \sum_{n=1}^{\infty} A_n J_1(\lambda_n s/a) e^{-\lambda_n^2 vt/a^2}. \quad (2.55)$$

Even though we've been dealing with Bessel function, the process of separation of variables is the same as we went through in Section 2.2.2 – if you can follow that discussion, you should be okay here, too. We just have one more step: finding the values of the constants A_n that will match our initial conditions. This part is similar to before, too, but does require a bit of advanced knowledge of Bessel functions, so watch carefully.

Our initial conditions, from equation (2.51), say

$$u_\phi(s, 0) = \sum_{n=1}^{\infty} A_n J_1(\lambda_n s/a) = \Omega s. \quad (2.56)$$

Just like sinusoidal functions, Bessel functions are orthogonal, although it looks a little different; it turns out that⁷

$$\int_0^a J_1(\lambda_n s/a) J_1(\lambda_m s/a) s ds = 0$$

if $m \neq n$. If $m = n$, then the result is

$$\int_0^a J_1^2(\lambda_n s/a) s ds = \frac{a^2}{2} J_2^2(\lambda_n)$$

(notice that's the second order Bessel function showing up on the right hand side).

We can use these results to solve for the A_n s. Multiply both sides of equation (2.56) by $s J_1(\lambda_m s/a)$ and integrate from 0 to a :

$$\sum_{n=1}^{\infty} A_n \int_0^a J_1(\lambda_n s/a) J_1(\lambda_m s/a) s ds = \Omega \int_0^a s^2 J_1(\lambda_m s/a) ds.$$

Orthogonality allows us to collapse the sum – only the m th term survives – and, performing the integrations, we get

$$A_n = \frac{2\Omega a}{\lambda_n J_2(\lambda_n)}.$$

Putting this result together with our general solution gives us, finally,

$$u_\phi(s, t) = 2\Omega a \sum_{n=1}^{\infty} \frac{J_1(\lambda_n s/a)}{\lambda_n J_2(\lambda_n)} e^{-\lambda_n^2 vt/a^2}. \quad (2.57)$$

Unlike our steady state solution, this one depends on the viscosity of the fluid – it controls how long it takes for the fluid to slow down. I've plotted the velocity in Figure 2.11, where you can see how the fluid behaves over time, but it might be useful to take a more physical look at this problem. I have a water bottle on my desk right now, half-filled with water, with a radius of about 4 cm. The viscosity of water, from Table 1.1, is $\nu = 0.01$ cm²/s. Taking, as we did before, the first term only in the sum,

$$u_\phi(s, t) \approx \frac{2\Omega a J_1(\lambda_1 s/a)}{\lambda_1 J_2(\lambda_1)} e^{-\lambda_1^2 vt/a^2}.$$

⁷See, for example, Arfken and Weber's *Mathematical Methods for Physicists*, Fourth Edition, p. 646.

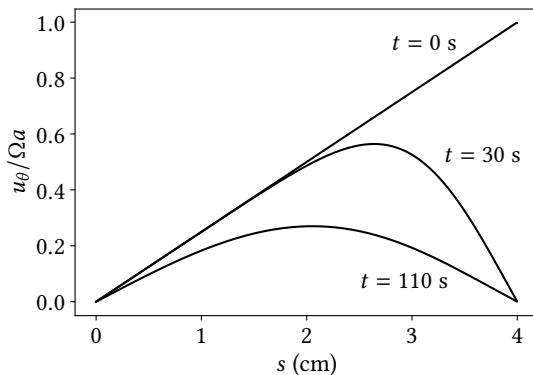


Figure 2.11: The spin down of fluid initially rotating like a solid body at three different times. In this case, I set $a = 4 \text{ cm}$ and $\nu = 0.01 \text{ cm}^2/\text{s}$ as in the water bottle example.

It's the exponential term we really want to look at; the characteristic time is

$$\tau = \frac{a^2}{\nu \lambda_1^2} \approx 110 \text{ s.}$$

That's a long time for my water bottle to slow down once I've rotated up to some speed. However, when I eyeball it, it looks like it takes about 25 s to come to a complete stop – much quicker! The thing we're missing in our analysis, of course, is the *bottom* of the bottle – it apparently plays an important role in the spin down of the water in the bottle.

2.3.4 The Line Vortex

Our last two examples involved fluid *inside* a cylinder; let's flip that around and look at the fluid *outside* a solid cylinder, rotating initially at some angular speed Ω (see Figure 2.12). As we've done a number of times in this chapter, we'll search first for a steady-state solution, and then look at time-dependence.

We once again have circular flow, and the steady-state is described by equation (2.48) and its general solution, equation (2.49),

$$u_\phi(s) = As + \frac{B}{s}.$$

Although the boundary condition is *also* the same – that $u_\phi(a) = \Omega a$ – the difference here comes from keeping the fluid velocity finite. Since the

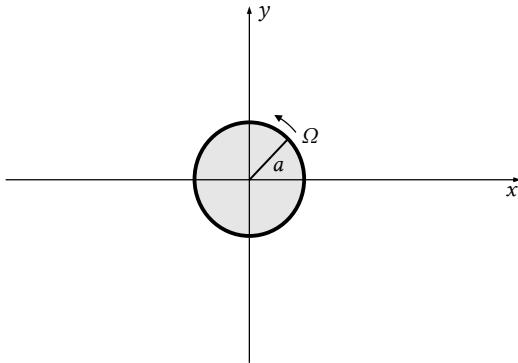


Figure 2.12: A solid cylinder rotating in a fluid.

fluid extends out to infinity, we set $A = 0$; with no fluid at $s = 0$, the second term won't be a problem. Fitting the boundary condition leads to our answer,

$$u_\phi(s) = \frac{\Omega a^2}{s}.$$

We've seen this kind of flow before – it's the line vortex from Example 1.8, with $k = \Omega a^2$. In fact, it's customary to write this flow as

$$u_\phi(s) = \frac{\Gamma_0}{2\pi s}, \quad (2.58)$$

where the constant Γ_0 is called the *circulation*. We'll explore the concept of circulation more later on in the book.

Now let's take the cylinder away. But we'll do it carefully – we'll let $a \rightarrow 0$ at $t = 0$, and the radius will shrink to zero instantaneously (kind of like how the boundary started moving in Section 2.2.3). That means equation (2.58) will give us the initial conditions, but now the evolution of the system is governed by equation (2.47),

$$\frac{\partial u_\phi}{\partial t} = \nu \left(\frac{\partial^2 u_\phi}{\partial s^2} + \frac{1}{s} \frac{\partial u_\phi}{\partial s} - \frac{u_\phi}{s^2} \right).$$

Before we solve this, I want to make a small change to this differential equation. Let's define a new variable,

$$\Gamma(s, t) = 2\pi r u_\phi(s, t). \quad (2.59)$$

Then, with a bit of work, the evolution equation becomes slightly simpler,

$$\frac{\partial \Gamma}{\partial t} = \nu \left(\frac{\partial^2 \Gamma}{\partial s^2} - \frac{1}{s} \frac{\partial \Gamma}{\partial s} \right), \quad (2.60)$$

and the initial conditions are now given by

$$\Gamma(r, 0) = \Gamma_0. \quad (2.61)$$

What about the boundary conditions? Well, as usual we want the flow to be finite everywhere for $t > 0$ (careful – right at $t = 0$ the flow u_ϕ is in fact infinite, a point we'll discuss in the next section). With our definition of Γ , that means we need

$$\Gamma(0, t) = 0 \quad \text{for } t > 0. \quad (2.62)$$

Now we're ready to solve this problem. Note that it's actually *scale-free* – there's nothing here (once the cylinder disappears) to provide any physical scale. That suggests we can seek a self-similar solution. As we did before in Section 2.2.4, we can define a new dimensionless variable,

$$\eta = \frac{s}{\sqrt{vt}}, \quad (2.63)$$

and assume that the solution is some function a of η alone,

$$\Gamma(s, t) = f(\eta).$$

I'll let you plug this into equation (2.60) and find the similarity solution yourself (Problem 2.9). The answer is

$$\Gamma(s, t) = \Gamma_0 \left(1 - e^{-s^2/4vt} \right),$$

so that the fluid velocity is

$$u_\phi(s, t) = \frac{\Gamma_0}{2\pi s} \left(1 - e^{-s^2/4vt} \right). \quad (2.64)$$

Let's explore this solution a bit. I've plotted the velocity in Figure 2.13 for a couple of different times so you can see how the speed decays. For large radii, $s \gg \sqrt{4vt}$, we can drop the exponential term in Equation 2.64, and the behaviour far from the origin still looks like a line vortex:

$$u_\phi \approx \frac{\Gamma_0}{2\pi s}.$$

However, at small radii, $s \ll \sqrt{4vt}$, we can expand the exponential and, keeping only the first two terms, get

$$u_\phi \approx \left(\frac{\Gamma_0}{8\pi vt} \right) s.$$

This looks like solid body rotation again, with $u_\phi \propto s$. Thus, at small radii, the flow is completely different; it undergoes solid body rotation and has *vorticity*. It's the role of vorticity we need to discuss next.

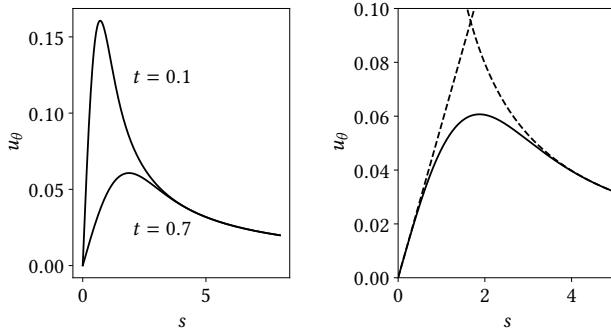


Figure 2.13: The decay of a line vortex. On the left plot, the fluid velocity is shown at two different times. On the right plot, just $t = 0.7$ is shown, along with the small- and large- radii behaviour shown as dashed lines. For convenience, I set $\Gamma_0 = 1$ and $v = 1$.

2.4 Transport of Vorticity

So far, all we've done is solve the Navier-Stokes equations for a variety of different situations. Actually, the variety hasn't been all that great – we've really only covered two kinds of flow, *plane parallel shear flow* and *circular flow*. In both cases, the flow direction is parallel to the flow dependence; in the case of plane parallel shear flow, we had the fluid moving in the x direction, but the speed changed along the y direction only. For circular flow, the fluid flowed in the ϕ direction but all dependence was along the radius s .

For flows of this type, the second term in the acceleration (equation (1.10)) – the so-called convective term – is zero:

$$(\mathbf{u} \cdot \nabla) \mathbf{u} = 0.$$

This not only makes the Navier-Stokes equations significantly easier (it eliminates their nonlinear nature, for example), it also allows for only one kind of *vorticity transport* as we'll see in a moment.

First, though, let's go back and take a look at two of our results: the impulsively moved boundary in Section 2.2.4 and the line vortex decay in Section 2.3.4. Both of these examples illustrated an interesting aspect concerning vorticity. For the impulsively moved boundary, the vorticity is (see Problem 2.10)

$$\omega = \frac{U}{\sqrt{\pi vt}} e^{-y^2/4vt}. \quad (2.65)$$

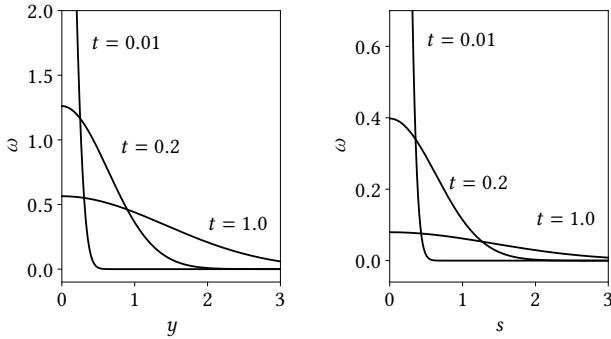


Figure 2.14: The vorticity ω at a few different times for the impulsively moved boundary (left) and the line vortex (right). In both cases an infinite amount of vorticity concentrated at one point spreads throughout the fluid.

As $t \rightarrow 0$, the vorticity becomes infinite at the boundary $y = 0$ but is zero everywhere else – this is called a *vortex sheet*. The line vortex is similar, with

$$\omega = \frac{\Gamma_0}{4\pi vt} e^{-s^2/4vt}. \quad (2.66)$$

This time, as $t \rightarrow 0$, the vorticity becomes infinite along $s = 0$, but zero elsewhere, so this is a *vortex line*. Figure 2.14 shows the two vorticities at various times. As you can see, the vorticity is strongly concentrated at either $y = 0$ or $s = 0$ at early times, but then becomes more “spread out” as time progresses. It’s this process of *vorticity diffusion* that I want to discuss.

In fact, there are two different ways that vorticity can change in a fluid. We can see this by taking the curl of both sides of the Navier-Stokes equations to get

$$\frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla) \omega = (\omega \cdot \nabla) \mathbf{u} + v \nabla^2 \omega \quad (2.67)$$

(this isn’t exactly straightforward to derive, but we’ll deal with it later in Section 3.1.3 for ideal fluids, and for now I don’t want any distractions). In two dimensional flow, like we’ve been dealing with in this chapter, it becomes a little simpler, since the first term on the right hand side will become zero – the vector ω will point in the z direction, so the term $\omega \cdot \nabla$ will have only a z derivative acting on \mathbf{u} , which of course only depends

on x and y . That means this vorticity equation looks like

$$\frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla) \omega = \nu \left(\frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2} \right).$$

We can now highlight the two ways that the vorticity can change: first by viscous *diffusion*, handled by the term with the viscosity in it, and second by *convection*, which is the second term on the left. By convection, I simply mean that the vorticity is transported through the fluid by the individual fluid elements themselves; in cases where the viscosity is zero, this is the only way to transport vorticity, and in that case each fluid element conserves its vorticity.

In all of the examples we've done so far, this convection term is zero, just like the convective acceleration is zero. That means the vorticity is only transported through the fluid by diffusion. In general, both mechanisms are at work, but of course those situations are significantly more difficult to deal with. One example that's actually solvable – with some difficulty and some numerical work – is detailed in Section 6.2.

Problems

Problem 2.1 – Poiseuille flow with gravity. In Section 2.2.1 we examined Poiseuille flow in one dimension, but without the force of gravity playing a role. Repeat our work to find the fluid velocity and pressure in the fluid, but this time include gravity in (a) the x direction (so $\mathbf{g} = [g, 0, 0]$) and (b) the z direction (so $\mathbf{g} = [0, 0, g]$). Comment on how the velocity changes. (*Note:* you can still call the pressure gradient along the x direction $P = -\partial p / \partial x$, but there could now also be addition dependence in the z direction.)

Problem 2.2 – Flow down an incline. Suppose fluid has been flowing down an inclined plane for a long time, as shown in Figure 2.15. The fluid has a height h above the plane, and above the fluid is air. Gravity points straight down, making an angle α with the y axis. What is the pressure and velocity of the fluid?

Hint: One of the boundary conditions is obvious, but the second is trickier. Assume the pressure at the surface of the fluid is at atmospheric pressure, and also that there is no tangential stress across the surface (otherwise it wouldn't be a stable surface); see equation (1.15).

Problem 2.3 – Poiseuille flow in two dimensions. Consider the rectangular pipe shown in Figure 2.16. Suppose fluid flows down the x -axis

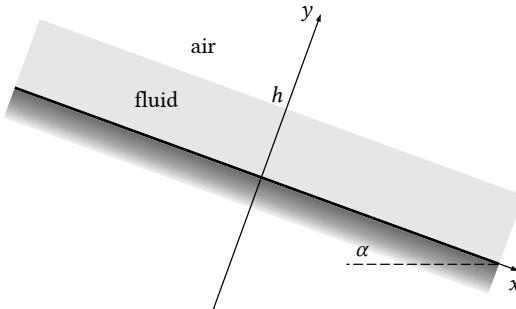


Figure 2.15: Fluid flows down an incline.

thanks to a constant pressure gradient

$$\frac{dp}{dx} = -P,$$

just as in the Poiseuille problem we discussed in Section 2.2.1. Find the velocity of the fluid within the pipe, and think about how to best visualize the flow.

Hint: The flow will now depend on z as well as y , and the differential equation will be nonhomogeneous, so you'll have to figure out a way to make it homogeneous. Think about the 1D solution we did earlier. This is a lengthy, difficult problem, so here's the final answer:

$$u(y, z) = \frac{P}{2\nu\rho} \left[(ay - y^2) + \sum_{n=1,3,5,\dots} \frac{8a^2}{\pi^3} \frac{1}{n^3} \frac{1}{\cosh(n\pi b/2a)} \sin(n\pi y/a) \cosh(n\pi z/a) \right]. \quad (2.68)$$

Problem 2.4 – Poiseuille flow in a cylinder. The physician Poiseuille originally studied the problem of blood flowing through blood vessels. We can model that with a Newtonian fluid flowing down a pipe of circular cross-section (radius a). If the pumping of the heart provides a constant pressure gradient within the blood vessel of $dp/dz = -P$, find the velocity of the fluid. Plot the velocity as a function of radius.

Problem 2.5 – Starting up Poiseuille flow. For one last Poiseuille flow problem, consider the 1D example of Section 2.2.1 again. This time, however, suppose that the fluid in the channel is at rest initially. At time $t = 0$,

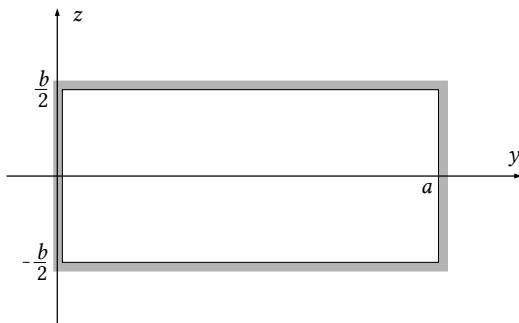


Figure 2.16: Fluid flows through a rectangular pipe.

the pressure gradient $dp/dx = -P$ starts instantaneously. How does the fluid respond? Find the velocity as a function of time, and plot it for a few different times. What does the flow look like for $t \gg h^2/v$?

Problem 2.6 – Fluid above an oscillating floor. Fluid lies above an infinitely rigid plane at $y = 0$. The plane is oscillating back and forth along the x -direction with velocity

$$u_{\text{plane}} = U \cos \omega t.$$

Find the velocity of the fluid,⁸ and plot it at some time t . How far above the plane does the oscillatory motion seem to affect the fluid?

Hint: Try a solution of the form $f(y)e^{i\omega t}$ and take the real part at the end.

Problem 2.7 – The incompressibility condition for circular flow. Show that the incompressibility condition is automatically satisfied if the flow is of the form

$$\mathbf{u} = u_\phi(s, t) \hat{\phi}.$$

Problem 2.8 – Fluid between two cylinders. Fluid lies between two concentric cylinders. The inner solid cylinder, of radius a , rotates with angular velocity Ω_a , and the outer hollow cylinder, of radius b , rotates with angular velocity Ω_b . Assuming the flow is steady, find the velocity of the fluid.

⁸This is Stokes' second problem.

Problem 2.9 – Similarity solution examined. Consider the decay of a line vortex, which we looked at in Section 2.3.4. The evolution of the vortex is given by Equation (2.60). Solve this equation for $\Gamma(s, t)$ by assuming a similarity solution of the form

$$\Gamma(s, t) = f(\eta),$$

where

$$\eta = \frac{s}{\sqrt{vt}},$$

and show that the velocity is given by equation (2.64). Plot the circulation $\Gamma(s, t)$ at two different times; scale your axis appropriately to show that the curve is self-similar.

Problem 2.10 – Vorticity of simple flows.

- (a) Find the vorticity of the fluid in the impulsively moved single plate example (Section 2.2.4) and show that it is given by equation (2.65).
- (b) Find the vorticity of the fluid in the line vortex example (Section 2.3.4) and show that it is given by equation (2.66).

Chapter 3

Ideal Fluids

3.1 Describing Ideal Flow

There are often situations where the viscosity of a fluid is unimportant – for example, in high Reynolds number flow. In those cases we can set the viscosity to zero and the fluid is called *inviscid*. If we also keep our flow incompressible (so that the density is constant throughout the fluid) then the fluid is *ideal*. Of course, real fluids always have *some* viscosity; however, as discussed in Section 1.6, in many cases an ideal fluid can be a good approximation to a real one. Keep in mind, however, that the boundary layer present in viscous fluids can never be described by an ideal fluid, so we'll miss some of the physics due to that.

Under these assumptions, the Navier-Stokes equation is usually called *Euler's equation*,

$$\boxed{\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla p + \mathbf{g}.} \quad (3.1)$$

The incompressibility condition,

$$\nabla \cdot \mathbf{u} = 0, \quad (3.2)$$

remains the same.

3.1.1 Static Fluids

The simplest solution to Euler's equation (and, in fact, the Navier-Stokes equation) is the trivial one: suppose the fluid is at rest, so that $\mathbf{u} = \mathbf{0}$ everywhere. Then equation 3.1 reduces to

$$\mathbf{0} = -\frac{1}{\rho}\nabla p + \mathbf{g},$$

or

$$\nabla p = \rho g.$$

If we take the direction of gravity to be down as usual, so that $\mathbf{g} = [0, 0, -g]$, this equation says

$$\frac{\partial p}{\partial x} = 0, \quad \frac{\partial p}{\partial y} = 0, \quad \frac{\partial p}{\partial z} = -\rho g.$$

The first two equations just say that the pressure p doesn't depend explicitly on x or y , and we can integrate the third to get

$$p = p_0 - \rho gz, \tag{3.3}$$

where p_0 is the integration constant. If our fluid has a "free surface" – that is, it's open to the atmosphere – at $z = 0$, then p_0 is the atmospheric pressure.

This result is simple and tells us what we already know from experience: as you go down in depth ($z < 0$) in a fluid, the pressure increases. Note, however, that we've assumed the density ρ stays constant; in some fluids (like the atmosphere) it's a bad assumption, while in others (like the ocean, at least for reasonable depths) it's okay.

3.1.2 Bernoulli's Principle

Since gravity is a conservative force, we can always write it in terms of the gradient of a scalar, so that

$$\mathbf{g} = -\nabla\Phi,$$

where Φ is called the gravitational potential. For example, setting $\Phi = gz$ gives us our usual \mathbf{g} pointing down along the z axis.

With this change, Euler's equation becomes

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla \left(\frac{p}{\rho} + \Phi \right). \tag{3.4}$$

It turns out that the second term on the left can be written as (see Problem 3.1)

$$(\mathbf{u} \cdot \nabla) \mathbf{u} = (\nabla \times \mathbf{u}) \times \mathbf{u} + \nabla \left(\frac{1}{2} \mathbf{u}^2 \right), \tag{3.5}$$

where $\mathbf{u}^2 = \mathbf{u} \cdot \mathbf{u}$; as usual, be careful of all the different us we deal with. With this substitution, and moving the gradient onto the right hand side, Euler's equation is now

$$\frac{\partial \mathbf{u}}{\partial t} + (\nabla \times \mathbf{u}) \times \mathbf{u} = -\nabla \left(\frac{p}{\rho} + \frac{1}{2} \mathbf{u}^2 + \Phi \right). \tag{3.6}$$

This doesn't look any better than the original form of Euler's equation, though. We can clean it up a bit by defining

$$H \equiv \frac{p}{\rho} + \frac{1}{2}\mathbf{u}^2 + \Phi,$$

which is sometimes called the "total head" or "energy head" of the flow. If we also assume the flow is *steady*, we then have

$$(\nabla \times \mathbf{u}) \times \mathbf{u} = -\nabla H. \quad (3.7)$$

One last step – take the dot product with \mathbf{u} for both sides:

$$\mathbf{u} \cdot [(\nabla \times \mathbf{u}) \times \mathbf{u}] = -\mathbf{u} \cdot \nabla H.$$

But now the left hand side is zero – the term in the square brackets has a direction perpendicular to \mathbf{u} , so the dot product with \mathbf{u} vanishes. So we're left with

$$(\mathbf{u} \cdot \nabla)H = 0. \quad (3.8)$$

As we learned back in Section 1.4, this means that the quantity H is *constant along a streamline* for steady flow. This is *Bernoulli's streamline theorem*.

If the fluid is also irrotational, so that $\nabla \times \mathbf{u} = 0$, equation (3.7) gives us the stronger statement

$$\nabla H = 0, \quad (3.9)$$

so that H is constant everywhere in the fluid.

Example 3.1 – A leaky bucket. A large tank of water, open to the atmosphere at the top, suddenly springs a leak near the bottom (see Figure 3.1). If the hole is 1.0 m below the free surface, what is the speed of the water as it comes out the hole?

This is a good case for Bernoulli's principle, since the flow is (approximately) steady – the surface will drop only slowly if the hole is small. We can therefore use the streamline theorem:

$$H = \frac{p}{\rho} + \frac{1}{2}\mathbf{u}^2 + gz = \text{constant.}$$

Presumably there exists a streamline that connects the free surface at the top of the tank with the hole (joining points 1 and 2 as shown in Figure 3.1). Then we can evaluate H at both points:

- Point 1 $\rightarrow H_1 = \frac{p_1}{\rho} + \frac{1}{2}\mathbf{u}_1^2 + gz_1$

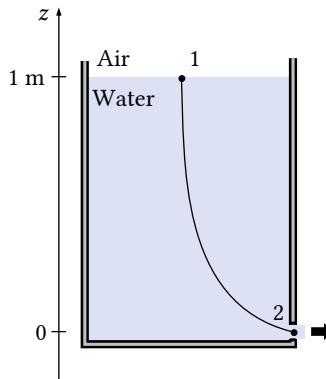


Figure 3.1: A large tank of water has sprung a leak. How fast is the water moving as it comes out of the hole?

- Point 2 → $H_2 = \frac{p_2}{\rho} + \frac{1}{2}\mathbf{u}_2^2 + gz_2$

But $p_1 = p_2 = p_0$ since both locations are open to the atmosphere. We'll also take $\mathbf{u}_1 = 0$, since the surface drops only very slowly. Finally, using the coordinate system shown in Figure 3.1, we have $z_2 = 0$ (and $z_1 = 1 \text{ m}$).

Setting $H_1 = H_2$ then gives us

$$\frac{p_0}{\rho} + gz_1 = \frac{p_0}{\rho} + \frac{1}{2}\mathbf{u}_2^2.$$

The pressure terms cancel, and we can rearrange for the speed of the water:

$$|\mathbf{u}| = \sqrt{2gz_1} \approx 4.4 \text{ m/s.}$$

3.1.3 The Vorticity Equation

Let's rewrite Euler's equation again, this time in terms of the vorticity. Going back to equation (3.7) and writing $\boldsymbol{\omega} = \nabla \times \mathbf{u}$, we have

$$\frac{\partial \mathbf{u}}{\partial t} + \boldsymbol{\omega} \times \mathbf{u} = -\nabla H.$$

Take the curl of both sides:

$$\frac{\partial \boldsymbol{\omega}}{\partial t} + \nabla \times (\boldsymbol{\omega} \times \mathbf{u}) = -\nabla \times \nabla H. \quad (3.10)$$

But the curl of a gradient is identically zero (see Problem 1.4), so the right hand side vanishes.

Now, there's a vector identity (another one!) that says, for two vectors \mathbf{F} and \mathbf{G} ,

$$\nabla \times (\mathbf{F} \times \mathbf{G}) = (\mathbf{G} \cdot \nabla) \mathbf{F} - (\mathbf{F} \cdot \nabla) \mathbf{G} + \mathbf{F}(\nabla \cdot \mathbf{G}) - \mathbf{G}(\nabla \cdot \mathbf{F}) \quad (3.11)$$

(see Problem 3.2). Using this in equation 3.10 above, with $\boldsymbol{\omega}$ replacing \mathbf{F} and \mathbf{u} replacing \mathbf{G} , gives us

$$\frac{\partial \boldsymbol{\omega}}{\partial t} + (\mathbf{u} \cdot \nabla) \boldsymbol{\omega} - (\boldsymbol{\omega} \cdot \nabla) \mathbf{u} + \boldsymbol{\omega}(\nabla \cdot \mathbf{u}) - \mathbf{u}(\nabla \cdot \boldsymbol{\omega}) = 0.$$

Of course, we're dealing with an ideal fluid, so $\nabla \cdot \mathbf{u} = 0$, and, since $\boldsymbol{\omega}$ is a curl, $\nabla \cdot \boldsymbol{\omega} = 0$ identically (see Problem 1.4 again for that one). So the fourth and fifth terms vanish, and we have

$$\frac{\partial \boldsymbol{\omega}}{\partial t} + (\mathbf{u} \cdot \nabla) \boldsymbol{\omega} = (\boldsymbol{\omega} \cdot \nabla) \mathbf{u}.$$

Finally, we can combine the two terms on the right hand side – that's the definition of the material derivative of $\boldsymbol{\omega}$ – and we have, at long last, the *vorticity equation*,

$$\frac{D\boldsymbol{\omega}}{Dt} = (\boldsymbol{\omega} \cdot \nabla) \mathbf{u}. \quad (3.12)$$

This equation will prove useful every once in a while for us. In particular, for a two dimensional flow, where

$$\boldsymbol{\omega} = [0, 0, \omega],$$

it's easy to see that the right hand side becomes zero, and we end up with

$$\frac{D\boldsymbol{\omega}}{Dt} = 0. \quad (3.13)$$

This means that the vorticity of each individual fluid element is conserved, a result we'll use later. Furthermore, if the flow is also steady, this becomes

$$(\mathbf{u} \cdot \nabla) \omega = 0, \quad (3.14)$$

and the vorticity is constant along streamlines in this case.

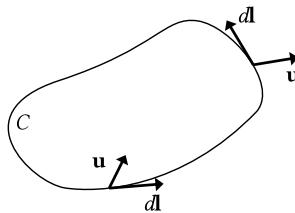


Figure 3.2: A closed curve C lies within the fluid region.

3.1.4 Circulation

Consider a closed curve C that lies in the fluid region as shown in Figure 3.2. The circulation Γ is defined as the line integral

$$\boxed{\Gamma = \oint_C \mathbf{u} \cdot d\mathbf{l}.} \quad (3.15)$$

Notice the role the dot product plays in the integral – as we go around the curve, only the fluid velocity that points in the direction of the curve is added, giving us the fluid that is circulating around that curve.

Example 3.2 – Line Vortex. We've explored the flow around a line vortex before – its vorticity in Example 1.8 and its generation in Section 2.3.4 – and we'll see it again later on. Now let's find the circulation around it.

The flow is given by

$$\mathbf{u} = \frac{\Gamma_0}{2\pi s} \hat{\phi},$$

and to calculate the circulation we'll use a circular path of constant radius R . The dot product in the integral becomes

$$\mathbf{u} \cdot d\mathbf{l} = \left(\frac{\Gamma_0}{2\pi s} \hat{\phi} \right) \cdot (ds \hat{s} + sd\phi \hat{\phi} + dz \hat{z}),$$

where we evaluate \mathbf{u} and $d\mathbf{l}$ along the curve, and get

$$\mathbf{u} \cdot d\mathbf{l} = \frac{\Gamma}{2\pi}.$$

The circulation is then

$$\Gamma = \oint_0^{2\pi} \frac{\Gamma_0}{2\pi} d\phi = \Gamma_0.$$

So the line vortex has *constant* circulation with strength given by Γ_0 .

We can write the defintion of the circulation in a different way using Stokes' theorem:

$$\Gamma = \oint_C \mathbf{u} \cdot d\mathbf{l} = \int_S (\nabla \times \mathbf{u}) \cdot d\mathbf{a}, \quad (3.16)$$

where S is the surface inside the closed curve with area vector $d\mathbf{a}$. Now, if $\nabla \times \mathbf{u} = \mathbf{0}$, then $\Gamma = 0$ as well, which means that the circulation is zero inside any irrotational flow. However, this is only true if the fluid is *everywhere* irrotational; in particular, if there is an object (like a cylinder or an airplane wing, both examples we'll be doing soon), Stokes' theorem fails and it *is* possible to have circulation in the fluid.

What if we suppose the closed curve actually moves with the fluid – so that the curve always constists of the same fluid elements? You can visualize this by pretending we put a bit of food colouring along the curve, so that as the fluid moves about and the curve changes shape, so too will the colouring – but it will always remain a close curve. We'll denote the curve now as $C(t)$ to indicate its dependence on time.

Now, *Kelvin's circulation theorem* says that the circulation Γ around $C(t)$ is *independent* of time – no matter how it changes as the fluid moves, Γ stays the same value. This is an important theorem, especially for understanding lift, so let's go through the proof.

Start with the time derivative of the circulation,

$$\frac{d\Gamma}{dt} = \frac{d}{dt} \left(\oint_{C(t)} \mathbf{u} \cdot d\mathbf{l} \right),$$

and move the derivative into the integral. Careful though – this is the *total* time derivative, and becomes the material derivative inside:

$$\frac{d\Gamma}{dt} = \oint_{C(t)} \frac{D\mathbf{u}}{Dt} \cdot d\mathbf{l}.$$

But from equation (3.4) we can write

$$\frac{D\mathbf{u}}{Dt} = -\nabla \left(\frac{p}{\rho} + \Phi \right),$$

so

$$\frac{d\Gamma}{dt} = \oint_{C(t)} \nabla \left(\frac{p}{\rho} + \Phi \right) \cdot d\mathbf{l} = - \left(\frac{p}{\rho} + \Phi \right) \Big|_{C(t)}.$$

We now have to evaluate the pressure, density, and gravitational potential along the curve $C(t)$ – but this is a closed curve and the integration takes us back to the starting point. Since all three quantities are single-valued functions, they have the same value at the start and end of the curve – so we get

$$\frac{d\Gamma}{dt} = 0. \quad (3.17)$$

This result is actually very general – the fluid could be viscous and could have holes; as long as $C(t)$ moves with the fluid the circulation along it will be constant.

3.1.5 The Surface of a Rotating Fluid

Let's end this section with a classic problem involving ideal fluids: the spinning water bucket example. In short, we'll fill a bucket with water and start rotating the bucket, letting it come to a steady state. We solved this problem already back in Section 2.3.2, where we found the steady state of the fluid was described by the velocity

$$\mathbf{u} = \Omega s \hat{\phi},$$

where Ω was the angular speed of the rotating boundary – the bucket in this case – and the rotation was about the z axis. Converting this to Cartesian coordinates gives us

$$\mathbf{u} = [-\Omega y, \Omega x, 0]. \quad (3.18)$$

Now, we *found* this velocity from the Navier-Stokes equation, and viscosity is important in getting the fluid into rotation in the first place, but once steady-state has been reached, the viscosity is no longer important and the fluid can be treated as ideal. We can thus examine the flow further using only Euler's equation rather than the full Navier-Stokes.

If you look at a good photograph of this, you'll see that the surface of the water is *curved* (Figure 3.3). What is the shape of this free surface? Well, the thing that all fluid elements along the free surface have in common is that they have the same pressure – namely, since they're at the surface, atmospheric pressure p_0 . Let's find the pressure in the water, then.

The x component of Euler's equation, with $\mathbf{g} = [0, 0, -g]$, is

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial x}.$$



Figure 3.3: A tank of water is smoothly spun up, causing the water to “climb” the sides of the tank.

But the flow is *steady*, and u doesn’t depend on x or z . Substituting in the fluid velocity in equation (3.18), this equation reduces to

$$\frac{\partial p}{\partial x} = \rho\Omega^2x. \quad (3.19)$$

Similarly, the v and w equations reduce to

$$\frac{\partial p}{\partial y} = \rho\Omega^2y \quad (3.20)$$

and

$$\frac{\partial p}{\partial z} = -\rho g. \quad (3.21)$$

We can now integrate to find the pressure $p(x, y, z)$. From equation (3.19) we get

$$p = \frac{1}{2}\rho\Omega^2x^2 + f(y, z),$$

where the function $f(y, z)$ is there since equation (3.19) is a *partial* derivative – we don’t just get a constant of integration, but a possible function of the other two variables. Integrating equation (3.20) gives

$$p = \frac{1}{2}\rho\Omega^2y^2 + g(x, z),$$

and integrating equation (3.21) gives

$$p = -\rho gz + h(x, y).$$

By inspection, it’s clear that the pressure must be

$$p(x, y, z) = \frac{1}{2}\rho\Omega^2(x^2 + y^2) - \rho gz + p_1,$$

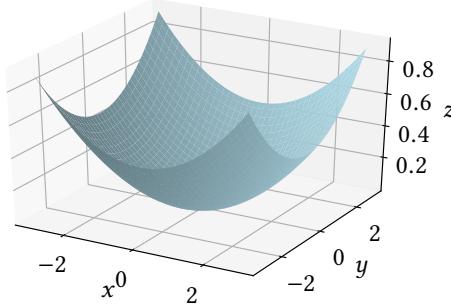


Figure 3.4: The free surface of the spinning bucket problem is a paraboloid.

where p_1 is a constant – it's the pressure at the origin where $x = y = z = 0$.

That's the pressure everywhere in the water. To find the shape of the free surface, we'll set $p = p_0$ and solve for the height z :

$$z = \frac{\Omega^2}{2g}(x^2 + y^2) + \left(\frac{p_1 - p_0}{\rho g} \right). \quad (3.22)$$

This is a *paraboloid* – see Figure 3.4 – and it matches the shape in the photograph in Figure 3.3.

3.2 The Velocity Potential and Stream Function

3.2.1 The Velocity Potential

For any irrotational flow, where

$$\nabla \times \mathbf{u} = 0,$$

a scalar function φ can be defined such that

$$\boxed{\mathbf{u} = \nabla \varphi.} \quad (3.23)$$

Then, since the curl of a gradient is zero, irrotationality is automatically satisfied. The quantity φ is called the *velocity potential*. In addition, from Stokes' theorem, we also have that

$$\oint \mathbf{u} \cdot d\mathbf{l} = \int (\nabla \times \mathbf{u}) \cdot d\mathbf{a} = 0,$$

so that the line integral of \mathbf{u} around any closed loop in the fluid is zero. This suggests we could also write the velocity potential as

$$\varphi(\mathbf{r}) = \int_{\mathcal{O}}^{\mathbf{r}} \mathbf{u} \cdot d\mathbf{l}, \quad (3.24)$$

where \mathcal{O} is an arbitrary point in the fluid. Incidentally, sorry for the notation – I’m using the Greek letter phi for three things: the gravitational potential Φ , the cylindrical coordinate ϕ , and now the velocity potential φ . I hope it’s not too confusing in context.

This might be familiar to you from electrostatics, where the electric field is also irrotational and we can define an electric potential. There’s a subtle point that frequently arises in fluid dynamics, though, so be careful. As long as the region is *simply connected* – that is, the fluid has no holes in it – the path between \mathcal{O} and \mathbf{r} doesn’t matter, and this leads to φ being a single-valued function. However, if the region is *multiply connected* – it has holes, regions where there is no fluid – the integral *could* depend on the path and φ will be a multivalued function of position. This can happen if the fluid is surrounding an object, something we’ll start to encounter more frequently. Let’s investigate this in more detail with two examples.

Example 3.3 – Flow past a stagnation point. Recall the flow past a stagnation point (from Examples 1.3 to 1.6), given by

$$\mathbf{u} = [\alpha x, -\alpha y, 0].$$

This is irrotational flow (but check to be sure!), and to find the velocity potential, we write

$$\frac{\partial \varphi}{\partial x} = u = \alpha x \quad \text{and} \quad \frac{\partial \varphi}{\partial y} = v = -\alpha y.$$

Integrating each term and combining gives

$$\varphi(x, y) = \frac{1}{2}\alpha(x^2 - y^2) + c,$$

where c is the integration constant. However, since it’s the fluid velocity, not the potential, that is the physically meaningful quantity, we can set the constant c to zero without losing anything – after all, we’ll take a derivative of φ to get the velocity, and the constant will end up going anyway.

Note that, in this example, φ is a single-valued function of x and y . That means that, for any point in the fluid, φ has a single value. This might not always be the case, as our next example will show.

Example 3.4 – Line vortex flow. Next, consider the flow

$$\mathbf{u} = \frac{\Gamma}{2\pi s} \hat{\phi}.$$

This is the flow around a line vortex, which we've also seen before. We have to be a bit more careful for this flow – it's irrotational (we showed that back in Problem 1.8), but not at the origin where it blows up. To fix this problem, we'll suppose there's *no* fluid there – maybe there's a cylinder of radius a there instead, covering up the problem area. That means the fluid domain is $s \geq a$, but it also means there's now a hole in the fluid domain; it's multiply connected.

The potential is found from equation (3.23) as usual, but in cylindrical coordinates it looks like

$$\frac{\partial \varphi}{\partial s} = u_s = 0, \quad \frac{1}{s} \frac{\partial \varphi}{\partial \phi} = u_\phi = \frac{k}{s}, \quad \text{and} \quad \frac{\partial \varphi}{\partial z} = u_z = 0.$$

Integrating this gives

$$\varphi(\phi) = k\phi. \tag{3.25}$$

But note that this isn't a single-valued function – φ has different values at the same point in space: $\varphi(0) = 0$, but $\varphi(2\pi) = 2\pi k$, and so on. We'll see later on that this is connected to circulation within the fluid.

Finally, if the fluid is both irrotational *and* incompressible, it must also satisfy the incompressibility condition,

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

If we rewrite this in terms of the potential, we get

$$\nabla \cdot \nabla \varphi = 0,$$

or

$$\boxed{\nabla^2 \varphi = 0.} \tag{3.26}$$

This is *Laplace's equation*; any irrotational, incompressible fluid must satisfy it.

3.2.2 Flow Past a Cylinder

Working with the velocity potential rather than the velocity itself is often easier; not only is a scalar function easier to work with than a vector

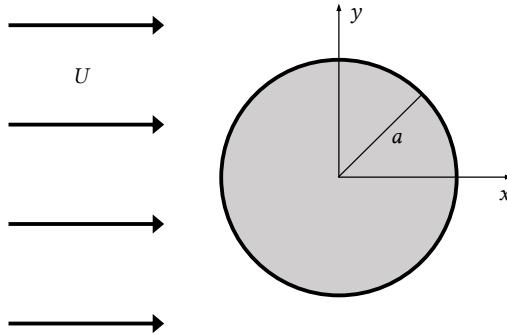


Figure 3.5: Fluid flows past the cylinder along the x -direction.

field, but Laplace's equation can be easier to solve than Euler's equation, which also contains the pressure of the fluid as an unknown. As an example of solving Laplace's equation, we'll find the ideal flow past a cylinder oriented perpendicular to the flow.

We'll start with a few basic assumptions. First, we'll treat this as two dimensional flow, which means the cylinder is effectively infinitely long. We'll put the cylinder, which has a radius a , along the z -axis, and the flow will be uniform in the x -direction infinitely far away from the cylinder (see Figure 3.5):

$$\mathbf{u}(\infty) = U \hat{\mathbf{x}}, \quad (3.27)$$

where U is the constant speed of the flow far away from the cylinder.

The other assumption we'll make is that the flow is irrotational. This seems like a leap to make, though, since we don't even know what the flow *is* yet. But remember the vorticity equation (equation 3.12) – for a steady two dimensional flow, the vorticity is conserved along streamlines. Since the vorticity at infinity, where the flow is *uniform*, is definitely zero, and every streamline starts and ends at infinity, it follows that the flow is everywhere irrotational.

Wait, is our flow *steady*? Yes, as long as we examine the problem from the point of view that the flow has been happening for a while and reached a steady state. In practice, it doesn't take long for the fluid to do this.

To find the fluid velocity around the cylinder, we'll solve Laplace's equation. It makes sense to use cylindrical coordinates here, given the symmetry of the boundary. In cylindrical coordinates, then, Laplace's

equation is

$$\frac{\partial^2 \varphi}{\partial s^2} + \frac{1}{s} \frac{\partial \varphi}{\partial s} + \frac{1}{s^2} \frac{\partial^2 \varphi}{\partial \phi^2} + \frac{\partial^2 \varphi}{\partial z^2} = 0. \quad (3.28)$$

Of course, we're assuming a two dimensional flow, so the z term we can safely ignore.

Our boundary conditions are straightforward to write down. First, as $s \rightarrow \infty$, we should have the uniform flow given by equation (3.27). But we need the potential rather than the velocity; following similar steps as in the examples above, the potential for uniform flow along the x -direction is

$$\varphi = Ux = Us \cos \phi, \quad (3.29)$$

where I've converted the Cartesian coordinates to cylindrical. Our first boundary condition, then, is that the flow must become this as $s \rightarrow \infty$.

Secondly, at $s = a$, we have the actual boundary. Unlike viscous flows, ideal fluids must "slip" along a boundary; in this case, that means we must have the fluid velocity at $s = a$ be purely in the $\hat{\phi}$ direction. In other words, we need $u_s = 0$; in terms of the potential (using the gradient in cylindrical coordinates), that's

$$\frac{\partial \varphi}{\partial s} = 0 \quad \text{at} \quad s = a. \quad (3.30)$$

We'll solve Laplace's equation using separation of variables. Let

$$\varphi(s, \phi) = S(s)\Phi(\phi).$$

Then equation (3.28) becomes

$$\Phi \frac{d^2 S}{ds^2} + \frac{\Phi}{s} \frac{dS}{ds} + \frac{S}{s^2} \frac{d^2 \Phi}{d\phi^2} = 0,$$

or, dividing by $S\Phi$ and multiplying by s^2 ,

$$\frac{s^2}{S} \frac{d^2 S}{ds^2} + \frac{s}{S} \frac{dS}{ds} = -\frac{1}{\Phi} \frac{d^2 \Phi}{d\phi^2}. \quad (3.31)$$

The left hand side of the equation is a function of s only, while the right hand side is a function of ϕ only; they must therefore both be equal to a constant. We'll call the this separation constant k^2 . The ϕ equation is then

$$\frac{d^2 \Phi}{d\phi^2} = -k^2 \Phi,$$

which has the solution

$$\Phi(\phi) = A \sin k\phi + B \cos k\phi.$$

We can apply our first boundary condition, equation (3.29), right away to eliminate the sine term, since we need only a cosine dependence as $s \rightarrow \infty$. Comparing the form of equation (3.29) with our solution, we furthermore must have $k = 1$. Thus

$$\Phi(\phi) = B \cos \phi. \quad (3.32)$$

The radial part of equation (3.31) (with $k = 1$) now reads

$$s^2 \frac{d^2 S}{ds^2} + s \frac{dS}{ds} - S = 0.$$

We've seen this differential equation before; it can be solved by trying a power-law solution of the form $S(s) = s^m$. The general solution is

$$S(s) = Cs + \frac{D}{s}.$$

But our second boundary condition, equation (3.30), was that the first derivative of the potential goes to zero at $s = a$. That means

$$\frac{dS}{ds} = \left(C - \frac{D}{s^2} \right) \Big|_{s=a} = 0,$$

and so $D = a^2 C$.

Combining the radial and angular equations gives us

$$\varphi(s, \phi) = S(s)\Phi(\phi) = C \left(s + \frac{a^2}{s} \right) \cos \phi,$$

where I absorbed the constants into C . One more comparison with our boundary condition, equation (3.29), tells us that $C = U$. So our solution to Laplace's equation is, finally,

$$\varphi(s, \phi) = U \left(s + \frac{a^2}{s} \right) \cos \phi. \quad (3.33)$$

That's the potential; what about the fluid velocity? No problem:

$$\mathbf{u} = \nabla \phi,$$

so

$$u_s(s, \phi) = U \left(1 - \frac{a^2}{s^2} \right) \cos \phi \quad (3.34)$$

and

$$u_\phi(s, \phi) = -U \left(1 + \frac{a^2}{s^2} \right) \sin \phi. \quad (3.35)$$

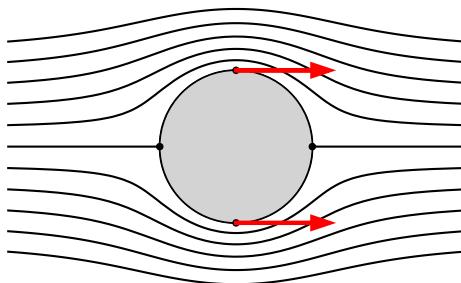


Figure 3.6: The streamlines for the flow around the cylinder. Note that there are stagnation points upstream and downstream of the cylinder ($\phi = 0$ and $\phi = \pi$), and the fluid has greatest velocity at the top and bottom ($\phi = \pi/2$ and $\phi = 3\pi/2$).

From \mathbf{u} , we can sketch streamlines, which are shown in Figure 3.6.

Let's examine the flow in a little more detail. Note that, as necessary, $u_s = 0$ at $s = a$. On the boundary, the flow is purely angular, with speed

$$u_\phi = -2U \sin \phi \quad \text{at} \quad s = a.$$

At $\phi = 0$ and $\phi = \pi$, then $u_\phi = 0$ and there is a stagnation point there (see Figure 3.6). There is a maximum speed at $\phi = \pi/2$ and $\phi = 3\pi/2$ of

$$u_{\phi,\max} = 2U \quad \text{at} \quad s = a.$$

What about the pressure in the fluid? Well, we could apply Euler's equation to find it, but Bernoulli's principle provides a shortcut. Since the flow is irrotational, $\nabla H = 0$ and $H = p/\rho + \frac{1}{2}\mathbf{u}^2 + gz = \text{constant}$ everywhere in the fluid. In our analysis, though, we'll neglect the gravity term; we're only looking at the (s, ϕ) dependence of the pressure here, and gravity will only impose an overall vertical pressure gradient.

Let's first evaluate H at infinity, where $\mathbf{u} = (U, 0, 0)$ and we'll label the pressure p_∞ . Then

$$H = \frac{p_\infty}{\rho} + \frac{1}{2}U^2.$$

Elsewhere in the fluid, it's

$$H = \frac{p(s, \phi)}{\rho} + \frac{1}{2}\mathbf{u}^2,$$

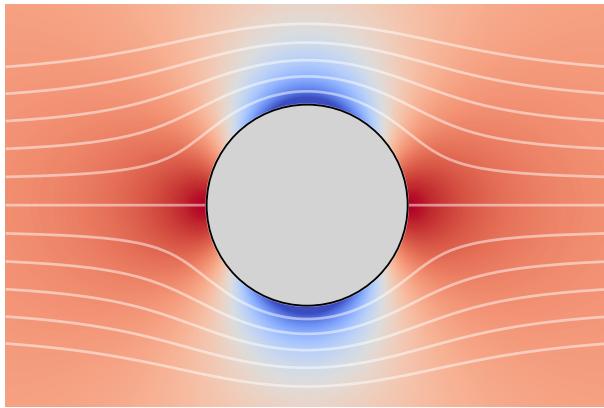


Figure 3.7: The pressure field of fluid flowing past a cylinder. The dark red regions are areas of high pressure, while the blue areas are low pressure. The streamlines are shown as well.

where

$$\mathbf{u}^2 = \mathbf{u} \cdot \mathbf{u} = u_s^2 + u_\phi^2 = U^2 \left(1 + \frac{a^4}{s^4} - 2 \frac{a^2}{s^2} \cos 2\phi \right)$$

(that last step required some algebra to clean up, though). Since H is constant, we have

$$\frac{p(s, \phi)}{\rho} + \frac{1}{2} U^2 \left(1 + \frac{a^4}{s^4} - 2 \frac{a^2}{s^2} \cos 2\phi \right) = \frac{p_\infty}{\rho} + \frac{1}{2} U^2.$$

Rearranging gives the pressure,

$$p(s, \phi) = p_\infty + \frac{1}{2} \rho \frac{a^2}{s^2} U^2 \left(2 \cos 2\phi - \frac{a^2}{s^2} \right). \quad (3.36)$$

Figure 3.7 shows the pressure around the cylinder; there is a region of high pressure at $\phi = 0$ and $\phi = \pi$, with low pressure at $\phi = \pi/2$ and $\phi = 3\pi/2$. Not surprisingly, this is opposite the fluid velocity – expected, since Bernoulli’s theorem holds here.

3.2.3 The Stream Function

Let’s take a break from thinking about irrotational flow for a moment, and instead consider flow that is incompressible (it’s important to note that the velocity potential φ doesn’t require incompressibility to describe flow, just irrotationality). We’ll impose the further restriction that it is also two

dimensional only – so $\mathbf{u} = [u(x, y, t), v(x, y, t), 0]$. In this case, we'll define the *stream function* ψ by

$$\boxed{\mathbf{u} = \nabla \times (\psi \hat{\mathbf{z}}).} \quad (3.37)$$

Why this particular function? Well, consider it in Cartesian coordinates,

$$u = \frac{\partial \psi}{\partial y} \quad \text{and} \quad v = -\frac{\partial \psi}{\partial x}. \quad (3.38)$$

With this definition, the incompressibility equation is automatically satisfied:

$$\nabla \cdot \mathbf{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = \frac{\partial}{\partial x} \left(\frac{\partial \psi}{\partial y} \right) + \frac{\partial}{\partial y} \left(-\frac{\partial \psi}{\partial x} \right) = 0,$$

since the order of the derivatives doesn't matter. Compare this with the velocity potential: it ensures irrotational flow, while the stream function ensures incompressible flow, at the cost of requiring it to be two dimensional. By the way, in cylindrical coordinates, this definition becomes

$$u_s = \frac{1}{s} \frac{\partial \psi}{\partial \phi} \quad \text{and} \quad u_\phi = -\frac{\partial \psi}{\partial s}. \quad (3.39)$$

You might be wondering why the stream function has that particular name – and it turns out the reason is one of its most useful features. Note that

$$(\mathbf{u} \cdot \nabla) \psi = u \frac{\partial \psi}{\partial x} + v \frac{\partial \psi}{\partial y} = \frac{\partial \psi}{\partial y} \frac{\partial \psi}{\partial x} + \left(-\frac{\partial \psi}{\partial x} \right) \frac{\partial \psi}{\partial y} = 0.$$

Remember what this means from Section 1.4 – that ψ is constant on streamlines; hence the name. If we know the stream function ψ , we'll now be able to easily plot the streamlines: just set ψ to a constant. Different values of the constant will give you different streamlines.

Example 3.5 – Flow past a stagnation point. Let's continue on from Example 3.3 and find the stream function for the flow

$$\mathbf{u} = [\alpha x, -\alpha y, 0],$$

which is incompressible (feel free to check) and obviously two dimensional. From $u = \partial \psi / \partial y$ we get

$$\frac{\partial \psi}{\partial y} = \alpha x,$$

which integrates to

$$\psi(x, y) = \alpha xy + f(x),$$

where $f(x)$ is some possible function of x . From $v = -\partial\psi/\partial x$, we end up with something similar after integrating,

$$\psi(x, y) = \alpha xy + g(y),$$

where again $g(y)$ is our integration “constant.” Now, comparing the two equations for ψ , it’s clear that we must have

$$\psi(x, y) = \alpha xy. \quad (3.40)$$

Streamlines for this flow can be found from setting ψ to a constant, which we’ll call k :

$$\psi = \alpha xy = k,$$

or

$$y = \frac{k}{\alpha x} = \frac{c}{x},$$

where $c = k/\alpha$. This exactly matches the equation we found back in Example 1.4 for the streamlines of a flow about a stagnation point.

Example 3.6 – Line vortex flow. For the line vortex flow, from Example 3.4 with

$$\mathbf{u} = \frac{\Gamma}{2\pi s} \hat{\phi},$$

the process to finding the stream function is similar, but we have to use cylindrical coordinates. From equation (3.39), we get

$$u_s = 0 = \frac{1}{s} \frac{\partial \psi}{\partial \phi} \quad \text{and} \quad u_\phi = \frac{\Gamma}{2\pi s} = -\frac{\partial \psi}{\partial s}.$$

From the first equation we just find that ψ has no ϕ dependence, and from the second, upon integrating, we get

$$\psi(s, \phi) = -\frac{\Gamma}{2\pi} \ln s.$$

Once again we can find the streamlines from the stream function by setting ψ to a constant (call it k again). Doing so and

solving for s gives

$$s = e^{-2\pi k/\Gamma} = \text{constant.}$$

This makes sense, since we know this is circular flow – the streamlines are circles of constant radius s .

Like the velocity potential, the stream function satisfies Laplace's equation as long as the flow is both incompressible and irrotational – although here we need the additional constraint of two dimensional flow as well. The proof is straightforward: start with the condition for irrotational flow, $\nabla \times \mathbf{u} = \mathbf{0}$, and substitute in the definition of the stream function from equation (3.37) to get

$$\nabla \times [\nabla \times (\psi \hat{\mathbf{z}})] = \mathbf{0}.$$

But we can expand the curl of a curl as

$$\nabla \times [\nabla \times (\psi \hat{\mathbf{z}})] = \nabla[\nabla \cdot (\psi \hat{\mathbf{z}})] - \nabla^2(\psi \hat{\mathbf{z}}) = \mathbf{0},$$

and, for two dimensional flow, the first term goes to zero, so we're left with

$$\boxed{\nabla^2\psi = 0.} \quad (3.41)$$

So both the velocity potential φ and the stream function ψ satisfy Laplace's equation for two dimensional, irrotational, incompressible flow. Although not every problem in fluid dynamics satisfies these restriction, many do, and we'll exploit this in the next chapter to investigate some interesting and complex situations.

Problems

Problem 3.1 – Yet more vector calculus. Show that, for any vector field \mathbf{v} , you can write

$$(\mathbf{v} \cdot \nabla)\mathbf{v} = (\nabla \times \mathbf{v}) \times \mathbf{v} + \nabla\left(\frac{1}{2}\mathbf{v}^2\right).$$

Problem 3.2 – Please, no more vector calculus. Show that, for any two vector fields \mathbf{F} and \mathbf{G} , you can write

$$\nabla \times (\mathbf{F} \times \mathbf{G}) = (\mathbf{G} \cdot \nabla)\mathbf{F} - (\mathbf{F} \cdot \nabla)\mathbf{G} + \mathbf{F}(\nabla \cdot \mathbf{G}) - \mathbf{G}(\nabla \cdot \mathbf{F})$$

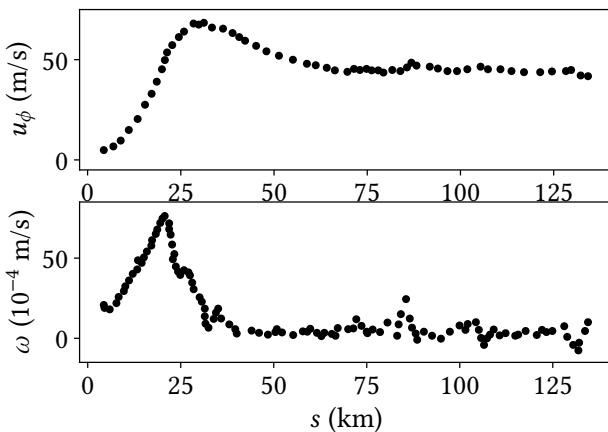


Figure 3.8: The azimuthal wind speed $u_\phi(s)$ and vorticity $\omega(s)$ for Hurricane Katrina. Data from Corbosiero, K. “Advanced Research WRF high resolution simulations of the inner core structures of Hurricanes Katrina, Rita, and Wilma (2005).” *Proceedings, 8th Annual WRF Users Workshop, Boulder, CO, NCAR*. 2007.

Problem 3.3 – The Rankine Vortex as a Model for Hurricanes.

A Rankine vortex is defined by the flow (in cylindrical coordinates)

$$\mathbf{u} = \begin{cases} \Omega s \hat{\phi} & \text{if } s < a \\ \frac{\Omega a^2}{s} \hat{\phi} & \text{if } s > a, \end{cases}$$

where Ω and a are constants (related to the wind speed and size of the “core,” respectively). It serves as a useful model for a number of weather and atmospheric conditions, such as hurricanes, mesocyclones, and tornadoes.

(a) Plot the wind speed as a function of distance, and calculate and plot the vorticity.

(b) To decide whether it is a useful model for hurricanes, I’ve provided azimuthal wind speed data and vorticity data for Hurricane Katrina, which devastated New Orleans and the surrounding area in 2005, in Figure 3.8 (you can extract this data using online tools such as PlotDigitizer). Use the Rankine vortex (RV) to model this hurricane; what values of Ω and a give you the best fit? Does the RV model the vorticity with those parameters as well? Can you make any conclusions about how well the model does (e.g., is it better in some regions versus others?)

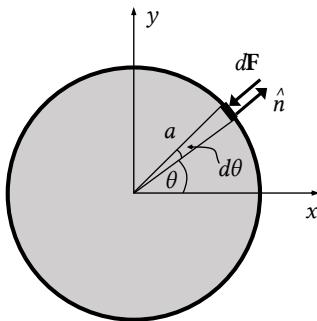


Figure 3.9: The force on a small angular section of the cylinder. The force is opposite the normal vector \hat{n} .

- (c) Calculate the pressure throughout the hurricane and find the difference in pressure between $r = 0$ (the eye of the hurricane) and $s \rightarrow \infty$ (outside the storm).
- (d) Finally, calculate the shape of the free surface (i.e., where the pressure is atmospheric). Plot the surface, and compare this with photos of Katrina – how does it look?

Problem 3.4 – The force on a cylinder. We've already calculated the pressure in the fluid around a cylinder in Section 3.2.2, so it's a short leap to find the total force exerted on the cylinder by the fluid.

Consider Figure 3.9: the force (per unit length of the cylinder) on a small angular section of size $a d\phi$ is

$$d\mathbf{F} = -p \hat{n}.$$

Using this, show the total force on the cylinder is zero. This is called *D'Alembert's paradox* – there's no drag on the cylinder, despite very obvious physical evidence to the contrary. Does it make sense that the fluid exerts no force at all on the cylinder as it flows past? Discuss.

Problem 3.5 – Uniform Flow. Consider two dimensional flow that is constant and flows in a direction at an angle α with respect to the x axis (see Figure 3.10).

- (a) Find the velocity potential $\varphi(x, y)$.
- (b) Find the stream function $\psi(x, y)$.

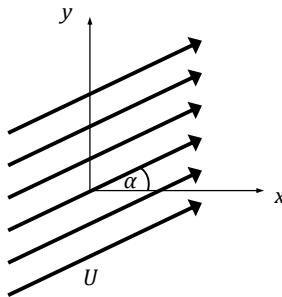


Figure 3.10: A constant flow that makes an angle α with the x axis.

Problem 3.6 – Line source and sink. Consider the two dimensional flow

$$\mathbf{u} = \frac{Q}{2\pi s} \hat{\mathbf{s}}, \quad (3.42)$$

where Q is a constant (you might want to compare the form of this flow to the line vortex). This is called a *line source* if Q is positive and a *line sink* if it's negative.

- (a) Produce a vector plot for the flow.
- (b) Find the velocity potential $\varphi(s, \phi)$.
- (b) Find the stream function $\psi(s, \phi)$.

Problem 3.7 – Plotting streamlines. Consider the two dimensional flow

$$\mathbf{u} = [2x + 3, -2y].$$

- (a) Show that the flow is irrotational and incompressible.
- (b) Find the stream function $\psi(x, y)$ and plot the streamlines.

Chapter 4

Potential Flow

4.1 The Complex Potential

In the previous chapter, we saw that an irrotational fluid can be described by the *velocity potential* φ , defined by

$$\mathbf{u} = \nabla \varphi. \quad (4.1)$$

If, instead, the fluid is both two dimensional and incompressible, it can be described by the stream function, ψ , defined via

$$\mathbf{u} = \nabla \times (\psi \hat{\mathbf{z}}). \quad (4.2)$$

Now, if we have a two dimensional fluid that is both irrotational *and* incompressible, then we can use either φ or ψ to describe it; in Cartesian coordinates, we have

$$u = \frac{\partial \varphi}{\partial x} = \frac{\partial \psi}{\partial y} \quad \text{and} \quad v = \frac{\partial \varphi}{\partial y} = -\frac{\partial \psi}{\partial x}. \quad (4.3)$$

For reasons we'll learn as we go, it's very convenient to define a *third* function, the *complex potential* $w(z)$:

$$w(z) = \varphi + i\psi. \quad (4.4)$$

Unlike the velocity potential and the stream function, the complex potential is a function of a *single* variable – the complex number z . That makes now a good time to review our knowledge of the complex plane.

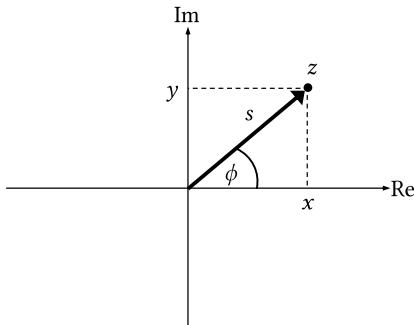


Figure 4.1: The complex plane showing the complex number $z = x + iy = s e^{i\phi}$.

4.1.1 The Complex Plane

Consider a complex number z :

$$z = x + iy. \quad (4.5)$$

We call x the *real part* of the complex number and y the *imaginary part*. This can be represented on a Cartesian coordinate system as shown in Figure 4.1; the x axis is the real axis and the y axis the imaginary. This is called the *complex plane*.

In addition to using Cartesian coordinates, we can also write the complex number z in polar coordinates; it should be clear from the figure that

$$z = x + iy = s \cos \phi + i \sin \phi.$$

Using Euler's famous formula,

$$e^{i\phi} = \cos \phi + i \sin \phi,$$

this becomes

$$z = s e^{i\phi}. \quad (4.6)$$

Some operations, like multiplying complex numbers, are easier in polar coordinates. Other operations are unique to complex numbers, such as taking the *complex conjugate* of z ; this means that every imaginary number i becomes the negative $-i$. The complex conjugate is denoted with a star:

$$z^* = x - iy = s e^{-i\phi}.$$

Although we can square a complex number as normal,

$$z^2 = (x + iy)^2 = x^2 - y^2 + 2ixy, \quad (4.7)$$

an operation that often comes up is the “complex square,”

$$|z|^2 \equiv z^*z = (x - iy)(x + iy) = x^2 + y^2. \quad (4.8)$$

Notice that the complex square will always give a real number.

Sometimes you’ll have a complex number in the denominator of a fraction, which makes it difficult to separate the real and imaginary parts of the complex number – something we’ll do pretty frequently. You can always get rid of the imaginary part in the denominator by multiplying by the complex conjugate:

$$\frac{1}{x + iy} = \frac{1}{x + iy} \frac{x - iy}{x - iy} = \frac{x - iy}{x^2 + y^2}.$$

So far, this is all pretty basic stuff, but we need at least one thing from the more advanced theory of complex analysis – we’ll be working with *complex functions* $f(z)$. Like complex numbers, complex functions can be separated into real and imaginary parts,

$$f(z) = f(x + iy) = A(x, y) + iB(x, y),$$

where $A(x, y)$ and $B(x, y)$ are real-valued functions. Now, a complex function is *analytic* if A and B are differentiable and satisfy the so-called *Cauchy-Riemann conditions*,

$$\frac{\partial A}{\partial x} = \frac{\partial B}{\partial y} \quad \text{and} \quad \frac{\partial A}{\partial y} = -\frac{\partial B}{\partial x}. \quad (4.9)$$

This means that the complex derivative exists, and can be written as¹

$$\frac{df}{dz} = \frac{\partial A}{\partial x} + i\frac{\partial B}{\partial x} = \frac{\partial B}{\partial y} - i\frac{\partial A}{\partial y}. \quad (4.10)$$

You might have noticed that the Cauchy-Riemann conditions look a lot like equation (4.3) – the relationship between the velocity potential and the stream function. This is the connection to fluid dynamics: by construction, the complex potential $w(z)$ is an analytic complex function, which means we can write the derivative of it as

$$\frac{dw}{dz} = \frac{\partial \varphi}{\partial x} + i\frac{\partial \psi}{\partial x} = u - iv.$$

(4.11)

¹This is obviously a drastically shortened and simplified look at complex analysis; for more, see e.g., *Mathematical Method for Physicists*, 4th edition, by Arfken and Weber, Chapter 6.

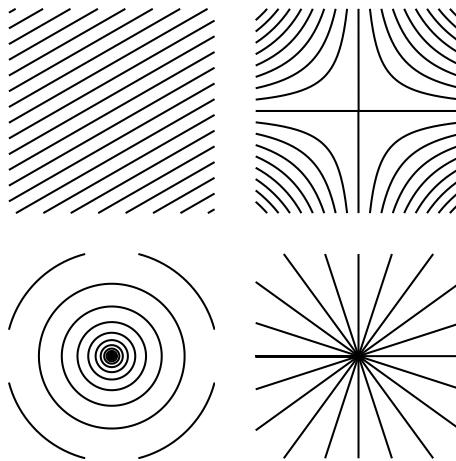


Figure 4.2: Four basic potential flows. Upper left: uniform flow at an angle α .

Upper right: flow about a stagnation point.

Lower left: a line vortex.

Lower right: a line source.

Thus the derivative of the complex potential is related to the speed of the fluid by

$$|\mathbf{u}| = \left| \frac{dw}{dz} \right| = \sqrt{\left(\frac{dw}{dz} \right)^* \left(\frac{dw}{dz} \right)}.$$

This gives us a powerful tool to examine some pretty complicated flows.

4.1.2 Basic Flows

Although we're restricted to two dimensional, incompressible, irrotational flows, the complex potential is nonetheless a useful description of fluid flow in a variety of cases. We'll start here with four basic flows, and will build up to fluid flow around a wing of an airplane, all the while using tools of complex analysis.

Example 4.1 – Uniform flow. To start, consider a simple flow that is uniform everywhere. We'll suppose it flows at an angle α with the x axis, so that the velocity is

$$\mathbf{u} = [U \cos \alpha, U \sin \alpha, 0]. \quad (4.12)$$

Problem 3.5 asked you to work out the details, but the velocity

potential in this case is

$$\varphi(x, y) = U(x \cos \alpha + y \sin \alpha) \quad (4.13)$$

and the stream function is

$$\psi(x, y) = U(y \cos \alpha - x \sin \alpha). \quad (4.14)$$

To find the complex potential, we'll start with equation (4.4) to get

$$w = \varphi + i\psi = U(x \cos \alpha + y \sin \alpha) + iU(y \cos \alpha - x \sin \alpha).$$

We can simplify this with a bit of rearranging:

$$w = U [(x + iy) \cos \alpha - i(x + iy) \sin \alpha].$$

But $z = x + iy$ so we have

$$w = Uz(\cos \alpha - i \sin \alpha),$$

or, using Euler's formula,

$$w(z) = Uze^{-i\alpha}. \quad (4.15)$$

This is the complex potential for uniform flow at an angle α . Note that it is a function only of z – all the x s and y s have disappeared, which is necessary for the complex potential.

Example 4.2 – Stagnation point flow. Let's return to a flow we've studied plenty already, the flow about a stagnation point,

$$\mathbf{u} = [\alpha x, -\alpha y, 0].$$

In Example 3.3 we found that the velocity potential was

$$\varphi(x, y) = \frac{1}{2}\alpha(x^2 - y^2),$$

and in Example 3.5 we found the stream function,

$$\psi(x, y) = \alpha xy.$$

The complex potential for this flow is

$$\begin{aligned} w &= \varphi + i\psi = \frac{1}{2}\alpha(x^2 - y^2) + i\alpha xy \\ &= \frac{1}{2}\alpha(x^2 - y^2 + 2ixy). \end{aligned}$$

If this doesn't look familiar, take a look at equation (4.7) – the term in the brackets is just z^2 , so the complex potential is

$$w(z) = \frac{1}{2}\alpha z^2. \quad (4.16)$$

Example 4.3 – A line vortex. Another familiar flow is the line vortex, covered in Examples 3.4 and 3.6. For reference, the velocity for this flow is

$$\mathbf{u} = \frac{\Gamma}{2\pi s} \hat{\phi},$$

and the velocity potential and stream function are

$$\varphi(s, \phi) = \frac{\Gamma\phi}{2\pi} \quad \text{and} \quad \psi(s, \phi) = -\frac{\Gamma}{2\pi} \ln s.$$

We'll construct the complex potential in the same way as above, but we'll have to use polar coordinates to build the complex number z . We have

$$w = \frac{\Gamma\phi}{2\pi} - i\frac{\Gamma}{2\pi} \ln s$$

and can pull out a factor of $-i\Gamma/2\pi$ to get

$$w = -\frac{i\Gamma}{2\pi} (i\phi + \ln s).$$

Now we'll have to use a bit of a trick: we can write $i\phi$ as

$$i\phi = \ln(e^{i\phi}),$$

and then combine the two natural logarithms to get

$$w = -\frac{i\Gamma}{2\pi} \ln(se^{i\phi}).$$

But in polar coordinates, $z = se^{i\phi}$, so we finally have

$$w(z) = -\frac{i\Gamma}{2\pi} \ln z. \quad (4.17)$$

Table 4.1: Four basic potential flows.

Flow	Velocity	Complex Potential
Uniform flow	$\mathbf{u} = [U \cos \alpha, U \sin \alpha]$	$w(z) = Uze^{-i\alpha}$
Stagnation point	$\mathbf{u} = [\alpha x, -\alpha y]$	$w(z) = \frac{1}{2}\alpha z^2$
Line vortex	$\mathbf{u} = \frac{\Gamma}{2\pi s} \hat{\phi}$	$w(z) = -\frac{i\Gamma}{2\pi} \ln z$
Line source	$\mathbf{u} = \frac{Q}{2\pi s} \hat{s}$	$w(z) = \frac{Q}{2\pi} \ln z$

Example 4.4 – A line source. For our final example, consider the flow about a *line source*,

$$\mathbf{u} = \frac{Q}{2\pi s} \hat{s}, \quad (4.18)$$

where Q is a constant. Problem 3.6 has some more information, but the velocity potential is

$$\varphi(s, \phi) = \frac{Q}{2\pi} \ln s, \quad (4.19)$$

while the stream function is

$$\psi(s, \phi) = \frac{Q\phi}{2\pi}. \quad (4.20)$$

The complex potential for this flow is

$$w(z) = \frac{Q}{2\pi} \ln z, \quad (4.21)$$

and you can work out the details yourself in Problem 4.2.

These four basic flows are shown in Figure 4.2 and summarized in Table 4.1; we'll be working with them a lot. However, one of the powerful things about the complex potential is that they can *add together*. This is possible because, since both φ and ψ are solutions to Laplace's equation, so too is the complex potential $w = \varphi + i\psi$. And since the Laplacian is a linear operator, any combination of solutions is a solution as well.

For example, suppose we combine a line source with a uniform flow (with angle $\alpha = 0$):

$$w(z) = Uz + \frac{Q}{2\pi} \ln z.$$

This complex potential is a perfectly valid description of *some* fluid flow –

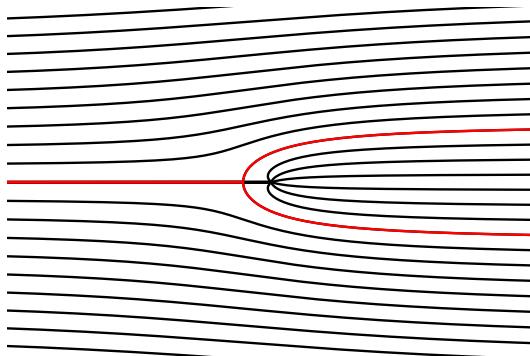


Figure 4.3: Streamlines for a flow that includes both uniform flow and a line source.

it's incompressible and irrotational and satisfies Laplace's equation. Now, whether or not this is flow that we actually *see* is up to experiment to test; in this case the flow represents fluid around an object called a *Rankine half body*.

We can investigate this flow further by finding the stream function and plotting streamlines. Remember that ψ is the imaginary part of the complex potential,

$$\psi = \text{Im}(w).$$

Finding the imaginary part isn't always simple for a general complex potential, although in this case it's not difficult and you should try it on your own. I'll discuss this a bit more in the next section; for now, the streamlines are plotted in Figure 4.3. It's not difficult to imagine how a flow like this might arise – the red streamline could be the boundary of an object in the fluid, and the flow is passing this object. This is because for ideal fluids the fluid must “slip” along a boundary (there's no no-slip condition for ideal fluids), meaning the boundary must be a streamline. This boundary condition will play an important role in our next example.

4.1.3 Method of Images

Consider a single line vortex, but we'll modify the flow in two ways: first, by moving it to a new position $z_0 = d$ along the x axis, and secondly by putting a wall at $x = 0$, effectively having the fluid in the domain $x > 0$ only. This is shown in Figure 4.4. Now, the first adjustment is easy to make, since moving the line vortex location just means shifting the origin in the opposite direction; in general, a line vortex at a location

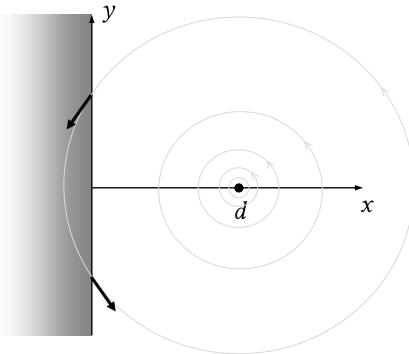


Figure 4.4: A line vortex a distance d from a wall.

$z_0 = x_0 + iy_0$ in the complex plane is given by

$$w(z) = -\frac{i\Gamma}{2\pi} \ln(z - z_0). \quad (4.22)$$

What about the wall? That's a little trickier, since it's apparent that the flow given in equation (4.22) doesn't satisfy the boundary conditions – we need the flow to slip along the wall. That means we require our flow at $x = 0$ to be along the y direction only, so that

$$u(0, y) = 0.$$

As shown in Figure 4.4, the flow is tangent to the streamlines, and doesn't point just along y at the wall. In fact, it looks like the fluid is either flowing *into* the wall or *out of* the wall, which doesn't make much sense physically. How can we adjust our complex potential to satisfy the boundary condition?

Well, never mind for a moment. Instead, consider a completely *different* problem: two vortexes, one at $z = d$ as above, and one at $z = -d$; there's no wall in this problem. The second vortex will have the same strength as the first, but will go in the opposite direction: $\Gamma_2 = -\Gamma_1$; see Figure 4.5. Since complex potentials add together, it's simple to write down $w(z)$ for this new flow:

$$w(z) = -\frac{i\Gamma}{2\pi} \ln(z - d) + \frac{i\Gamma}{2\pi} \ln(z + d)$$

(notice the plus sign – the second term is for the vortex that rotates in the opposite sense as the first). We can simplify this a bit using log rules to

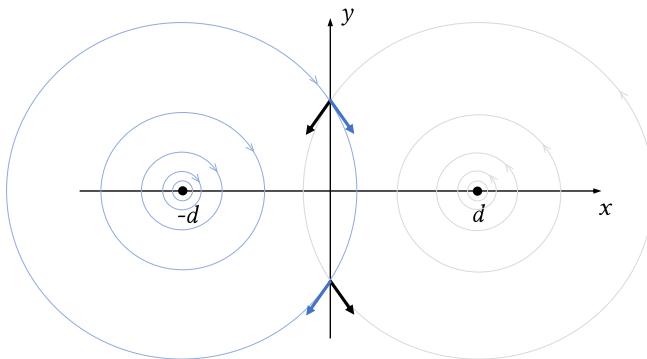


Figure 4.5: Two line vortices combine to create a flow that slips along the wall.

get

$$w(z) = -\frac{i\Gamma}{2\pi} \ln\left(\frac{z-d}{z+d}\right). \quad (4.23)$$

Now, *this* flow *does* satisfy the boundary condition – the two flows from each vortex combine at the wall and the x components of the velocity end up cancelling, giving components along the y direction only. In other words, this flow slips along the boundary; you can prove this mathematically if you want by trying Problem 4.6.

By adding a second vortex, we've satisfied the boundary conditions of the original problem; this is a technique called method of images, which you might know from electrostatics. Of course, in the original problem, we had only *one* vortex, not two – but that's okay, since our second vortex is covered up by the wall. It's not really there, and our solution is only good for the domain $x > 0$.

Method of images works because our fluid satisfies Laplace's equation, and solutions to Laplace's equation are *unique*: if a complex potential is a solution to Laplace's equation *and* satisfies the boundary conditions of the problem, then it's *the* solution.²

One more thing before we move on. What does the flow in $x > 0$ actually look like? Figure 4.5 gives streamlines for each vortex superimposed, not combined properly. To find the streamfunction, we need the stream function,

$$\psi = \text{Im}(w).$$

²Griffiths has an excellent discussion of uniqueness theorems and Laplace's equation in *Introduction to Electrodynamics*, 4th edition, Chapter 3; as usual there's much more to say about the matter.

As I mentioned above, it's not always easy to find the imaginary part of the complex potential, particularly when logarithms are involved. I'll walk you through a general method for doing so, and we'll start with the polar form of the complex number z :

$$z = se^{i\phi}.$$

Here, s is often called the *modulus*, and is the complex absolute value of z ,

$$s = |z| = \sqrt{z^* z},$$

and ϕ is called the argument. Now, if we just have the logarithm of a complex number, that's easy enough to split into real and imaginary parts, and we've already seen the reverse of it in Example 4.3:

$$\ln z = \ln s + i\phi.$$

Note that the real part of $\ln z$ is the logarithm of the modulus s and the imaginary part is the argument ϕ . Careful, though – in fact, I could add a multiple of 2π to ϕ and still get the same $\ln z$, so there's actually an infinite number of possibilities for the imaginary part of $\ln z$. Writing it as I did means $\ln z$ is called the *principal value*. This is something we won't have to worry about much, but I wanted to mention it for completeness.

If we have something more complicated, say the logarithm of a complex function $f(z)$, the idea is the same,

$$\ln [f(z)] = \ln |f(z)| + i \operatorname{Arg}[f(z)].$$

Now, it might not be simple to find the argument of the function $f(z)$ – you'll have to split it into its real and imaginary parts,

$$f(z) = A(x, y) + iB(x, y),$$

and then find the argument from

$$\operatorname{Arg}[f(z)] = \operatorname{Arg}[A(x, y) + iB(x, y)] = \tan^{-1} \left[\frac{B(x, y)}{A(x, y)} \right].$$

Things can once again get complicated in dealing with the arctan function, but luckily we won't have to worry about this much either. The modulus at least is normally not a problem, since

$$|f(z)| = \sqrt{f^*(z)f(z)}. \quad (4.24)$$

In the case of the vortex beside the wall, we can now write

$$w(z) = -\frac{i\Gamma}{2\pi} \ln \left(\frac{z-d}{z+d} \right) = -\frac{i\Gamma}{2\pi} \left[\ln \left| \frac{z-d}{z+d} \right| + i \operatorname{Arg} \left(\frac{z-d}{z+d} \right) \right]$$

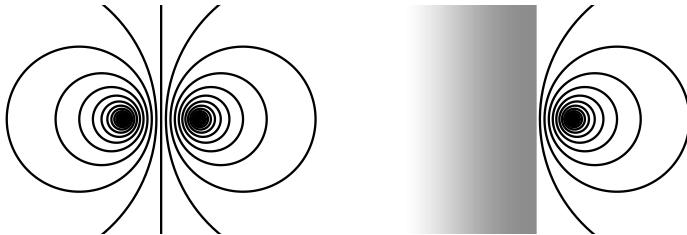


Figure 4.6: The streamlines for a line vortex a distance d from a wall. The left side shows both vortexes, while the right side shows just the real vortex beside the wall.

or

$$w(z) = \frac{\Gamma}{2\pi} \operatorname{Arg} \left(\frac{z-d}{z+d} \right) - i \frac{\Gamma}{2\pi} \ln \left| \frac{z-d}{z+d} \right|,$$

and we can identify the stream function as the imaginary part,

$$\psi(x, y) = \frac{\Gamma}{2\pi} \ln \left| \frac{z-d}{z+d} \right|. \quad (4.25)$$

Careful, though – there are still complex numbers z in that equation for ψ , which should be a real function. However, because the complex absolute value is there, it's guaranteed to be real – if you work it out (it's a bit of algebra but not too hard), you get

$$\psi(x, y) = -\frac{\Gamma}{2\pi} \ln \sqrt{\frac{x^2 + y^2 - 2xd + d^2}{x^2 + y^2 + 2xd + d^2}}. \quad (4.26)$$

From the stream function you can plot the streamlines of the flow as shown in Figure 4.6; notice that one streamline lies directly along the wall.

4.1.4 The Circle Theorem

Take a close look at Figure 4.6 – although the circular streamlines are no longer *concentric*, they are still circular. That suggests that this complex potential can also work for a line vortex within a *circular* boundary (say, a cylindrical bucket filled with fluid) – as long as the boundary matches one of the streamlines, our boundary condition will be satisfied and the fluid will slip along the circular wall.

Consider the situation shown in Figure 4.7. We'll put the origin of our coordinate system at the centre of the cylindrical bucket (of radius a)

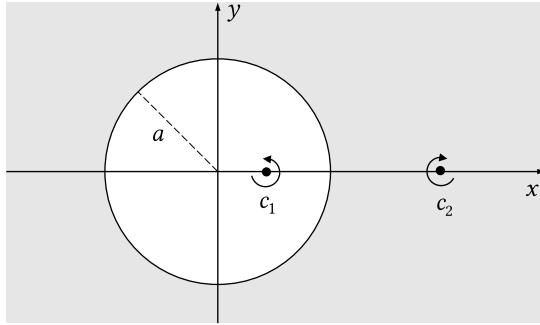


Figure 4.7: A line vortex in a circular bucket can be modelled with a mirror image outside the bucket.

and orient the x axis so that the line vortex of strength Γ is located on it at $x = c_1$. To ensure a streamline matches the boundary of the cylinder, we'll need a mirror image of the vortex (with opposite circulation $-\Gamma$) on the x axis – outside the cylinder. The question is *where* do we put it?

Take a look at the circles on the left side of Figure 4.6 again – they're a well-known system called *coaxal circles*. The members of a coaxial system are described by the equation

$$x^2 + y^2 + 2\lambda x + k = 0, \quad (4.27)$$

with different values of λ giving different circles. These two sets of circles have *limiting points* at $x = \pm\sqrt{-k}$ – that's where they converge to a circle of radius zero. In our problem here, the positions of the two line vortices at $x = c_1$ and $x = c_2$ would be the limiting points. From equation (4.27) you can derive a relationship between the radius a of one of the circles and the two limiting points; it turns out to be

$$a^2 = c_1 c_2.$$

If we have a vortex at $x = c_1$ inside the cylinder, placing a mirror vortex at

$$c_2 = \frac{a^2}{c_1} \quad (4.28)$$

will give streamlines that match the boundary. The complex potential of a vortex in a cylindrical bucket a distance c from the centre is therefore

$$w(z) = -\frac{i\Gamma}{2\pi} \ln(z - c) + \frac{i\Gamma}{2\pi} \ln(z - a^2/c) \quad (4.29)$$

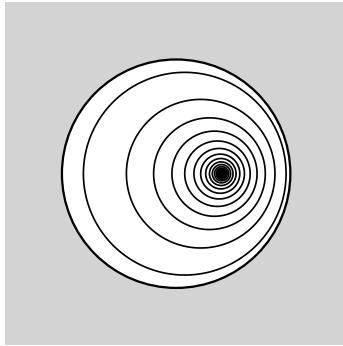


Figure 4.8: The streamlines of a line vortex within a circular cylinder.

(I've dropped the subscript “1” from c since we don't need c_2 anymore), and the streamlines are shown in Figure 4.8.

The idea of placing a vortex in a flow to get a circular streamline is very general – and powerful. The Milne-Thompson Circle Theorem gives us a way to add a circular streamline to any irrotational, incompressible flow, and is given by the following:

Milne-Thompson Circle Theorem. Consider a flow described by the complex potential

$$w = f(z),$$

and any singularities of $f(z)$ (in other words, where it goes to infinity) occur in the region $|z| > a$. Then the new complex potential

$$w = f(z) + f^*(a^2/z^*) \quad (4.30)$$

describes a flow that has the same singularities as $f(z)$ in $|z| > a$ and has the circle $|z| = a$ as a streamline.

This is straightforward to prove. The circle $|z| = a$ can be written as

$$|z|^2 = z^* z = a^2,$$

or

$$z^* = \frac{a^2}{z}.$$

That means that, on the circle itself,

$$w = f(z) + f^*(a^2/z^*) = f(z) + f^*(z).$$

But the function $f(z) + f^*(z)$ is a *real* function; suppose we write $f(z) = A(x, y) + iB(x, y)$, in which case it's easy to see that

$$f(z) + f^*(z) = A(x, y) + iB(x, y) + A(x, y) - iB(x, y) = 2A(x, y),$$

and the imaginary part has cancelled. If $w(z)$ is a *real* function on the circle, then its imaginary component must be zero:

$$\psi(x, y) = \text{Im}(w) = 0.$$

Never mind the zero – the important thing is to see that ψ is *constant* along the circle. That means the circle is a streamline!

What about the singularity part? Well, any point that is originally outside the circle in $f(z)$ actually ends up *inside* the circle in $f^*(a^2/z^*)$, so no new singularities outside $|z| = a$ are created – that means the new flow has the same singularities as the original flow.

4.2 Flow Past Objects

4.2.1 Flow Past a Cylinder

As a simple example of the Circle Theorem, consider starting with uniform flow with angle $\alpha = 0$, given by

$$w(z) = Uz.$$

Using the Circle Theorem we get

$$w(z) = Uz + \left[U \frac{a^2}{z^*} \right]^*$$

or

$$w(z) = U \left(z + \frac{a^2}{z} \right). \quad (4.31)$$

This is the complex potential for a flow that is uniform at infinity but has a circular streamline centred at the origin with radius a ; see Figure 4.9. The streamlines outside the circular streamline might look familiar if you recall Section 3.2.2, and we'll examine them closely in a moment. But what about the streamlines interior to the circle? They're similar to the streamlines for a *doublet* flow, in which $w \propto z^{-1}$, but that doesn't really

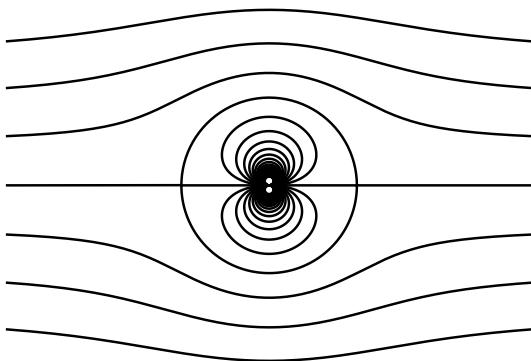


Figure 4.9: The streamlines for the flow generated by applying the Circle Theorem to a uniform flow.

matter – in fact, if we “cover up” the fluid interior to $|z| = a$ with, say, a circular cylinder, then what we’ve done is find the complex potential for flow around that cylinder.

Does our solution match the one we found earlier in Section 3.2.2? We can find the velocity potential and stream function easily enough (see Problem 4.11); they are

$$\varphi(s, \phi) = U \left(s + \frac{a^2}{s} \right) \cos \phi \quad (4.32)$$

and

$$\psi(s, \phi) = U \left(s - \frac{a^2}{s} \right) \sin \phi. \quad (4.33)$$

We can also find the velocity of the flow, either from the potential or stream function, or from the derivative of the complex potential directly using equation (4.11); either way, we get

$$u_s(s, \phi) = U \left(1 - \frac{a^2}{s^2} \right) \cos \phi \quad (4.34)$$

and

$$u_\phi(s, \phi) = U \left(1 + \frac{a^2}{s^2} \right) \sin \phi. \quad (4.35)$$

These do indeed match the flow we found earlier – by much more laborious means (solving Laplace’s equation). It’s hard to overstate just how powerful the Circle Theorem can be; we found in a few lines what took us four pages of solving a partial differential equation before.

But we can do more in this case. It's trivial to add an angle α to the uniform flow, for example; in that case we start with the complex potential

$$w(z) = Uze^{-i\alpha}$$

and get, using the Circle Theorem,

$$w(z) = U \left(ze^{-i\alpha} + \frac{a^2}{z} e^{i\alpha} \right). \quad (4.36)$$

We could also add a line vortex, placed at the origin, to our flow. This wouldn't change the fact that there's a circular streamline at $|z| = a$, since the line vortex has circular streamlines, but would change the flow u_ϕ at the surface of the cylinder – in effect, we're adding *circulation* to the flow, as we'll see. Remember, complex potentials add together, so the flow with a line vortex is simply

$$w(z) = U \left(ze^{-i\alpha} + \frac{a^2}{z} e^{i\alpha} \right) - \frac{i\Gamma}{2\pi} \ln z. \quad (4.37)$$

Streamlines with circulation in the flow are shown in Figure 4.10. Something interesting happens as the circulation Γ increases: notice that the two stagnation points, normally at opposite ends of the cylinder, come closer together as the circulation increases, until

$$\Gamma = \pm 4\pi U a,$$

when they combine to form just one stagnation point. If the circulation is greater than this, they disappear completely – we now have fluid circulating completely around the cylinder, as you can see by the streamlines in the lower right part of Figure 4.10 (note that the circulation is negative in the Figure, so that the fluid circulates clockwise around the cylinder; we'll see why I chose this shortly). You can explore this flow more in Problem 4.11.

Back in Problem 3.4 you may have calculated the force of the fluid on a circular cylinder for flow without circulation – it turned out to be zero, a somewhat surprising result called d'Alembert's paradox. Because we're dealing with ideal fluids and the flow is symmetric, the fluid is unable to push on the cylinder. However, this result changes when circulation is added to the flow. Problem 4.12 can walk you through the details of the calculation, but nothing changes for the x component of force – the fluid still doesn't push on the cylinder in the direction of the oncoming flow – but there *is* a force in the y direction, perpendicular to the uniform flow:

$$F_y = -\rho U T. \quad (4.38)$$

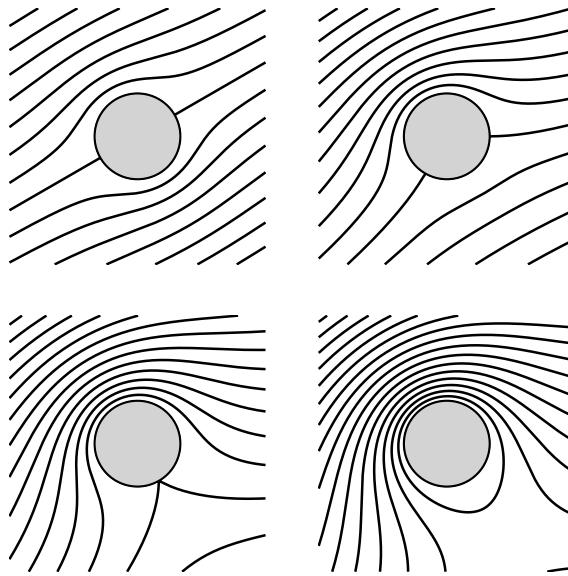


Figure 4.10: Streamlines for flow around a circular cylinder at an angle $\alpha = \pi/6$.
 Upper left: circulation $\Gamma = 0$. Upper right: circulation $\Gamma = -2\pi Ua$.
 Lower left: circulation $\Gamma = -4\pi Ua$. Lower right: circulation $\Gamma = -6\pi Ua$.

This force is due entirely to the circulation around the cylinder, and if Γ is negative, as the examples in Figure 4.10 show, then the force is upward: the fluid provides *lift* on the cylinder. Of course, the wings of an airplane aren't at all circular in cross section, but this essential idea – that circulation around the object is necessary for lift – will return when we discuss proper aerofoils.

4.2.2 Conformal Mapping

Suppose we have some general complex potential $w(z) = \varphi + i\psi$. Our goal is to create a *mapping* or *transformation* between the complex plane $z = x+iy$ and a second complex plane, which I'll write with capital letters: $Z = X+iY$. We'll perform this transformation with the function $f(z)$ such that

$$Z = f(z). \quad (4.39)$$

We'll assume $f(z)$ is an *analytic* complex function, meaning it's differentiable, and that it has an inverse given by

$$z = F(Z) \quad (4.40)$$

(and $F(Z)$ is also an analytic function).

The complex potential is transformed as well – in the Z complex plane it takes on the form

$$W(Z) = w[F(Z)]. \quad (4.41)$$

Now, we can split $W(Z)$ into its real and imaginary parts,

$$W(Z) = \Phi(X, Y) + i\Psi(X, Y). \quad (4.42)$$

Here's the amazing thing: because W will be an analytic function of Z , Φ and Ψ will satisfy the Cauchy-Riemann conditions (equation 4.9), and therefore

$$\mathbb{U}(X, Y) = \frac{\partial \Phi}{\partial X} = \frac{\partial \Psi}{\partial Y} \quad \text{and} \quad \mathbb{V}(X, Y) = \frac{\partial \Phi}{\partial Y} = -\frac{\partial \Psi}{\partial X} \quad (4.43)$$

represent a possible fluid flow – that is, the velocity $\mathbb{U}\hat{X} + \mathbb{V}\hat{Y}$ is an irrotational, incompressible flow that satisfies Laplace's equation.

This might seem like a lot to take in at first, but our goal is to use a transformation to map our flow around a circular cylinder to other possible shapes – including, eventually, a realistic wing shape. Before we get there, though, let's take a look at a simple example.

Example 4.5 – A Simple Conformal Map. Consider the map

$$Z = f(z) = \frac{z^2}{5} + 2,$$

which has the inverse

$$z = F(Z) = \sqrt{5(Z - 2)}.$$

We'll start with the complex potential for uniform flow, $w(z) = Uz$; the left side of Figure 4.11 shows the streamlines (blue) as well as lines of constant velocity potential (red). For uniform flow, these lines are straight and meet at right angles.

Under the map, the complex potential in the Z complex plane becomes

$$W(Z) = w[F(Z)] = U\sqrt{5(Z - 2)};$$

streamlines for this transformed complex potential are shown on the right panel of Figure 4.11. The original streamlines are mapped

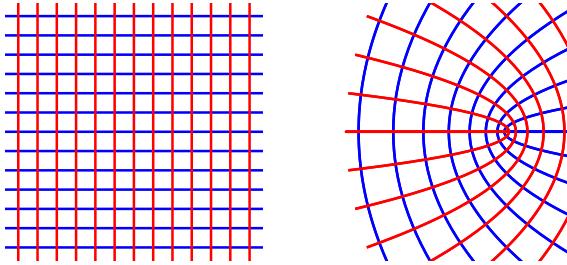


Figure 4.11: The effect of the conformal map $f(z) = z^2/5 + 2$ on uniform flow. The blue lines are the streamlines $\psi = \text{constant}$ and the red lines are lines of constant velocity potential φ . Left: the original uniform flow, $w(z) = Uz$, in the z complex plane. Right: the flow under the conformal map $W(Z)$ in the Z complex plane.

to new streamlines (and similarly for the lines of constant φ), and are no longer straight – but notice that they do still cross at right angles. This is why the mapping is called conformal – the transformation preserves local angles.

Suppose we look at just one point in the z complex plane, say

$$z_0 = 1 + i.$$

The complex potential at that point is $w(z_0) = U(1 + i)$. In the Z complex plane, z_0 is mapped to

$$Z_0 = \frac{2i}{5} + 2,$$

but the complex potential $W(Z_0)$ has the same value as $w(z_0)$:

$$W(Z_0) = U\sqrt{5(2i/5 + 2 - 2)} = U\sqrt{2i} = U(1 + i).$$

When choosing a particular map, there are two things to be careful about. First, consider the derivative of $W(Z)$. Writing it as $W(Z) = w(z)$ where $z = F(Z)$, we can use the chain rule to get

$$\frac{dW}{dZ} = \frac{dw}{dz} \frac{dz}{dZ}.$$

But recall that $dw/dz = u - iv$, so this says

$$\mathbb{U} - i\mathbb{V} = \frac{u - iv}{dZ/dz} = \frac{u - iv}{f'(z)}, \quad (4.44)$$

using $Z = f(z)$. If we want the streamlines in the new Z complex plane to match the original streamlines in the z plane at infinity – far away from the object the fluid is flowing past – then we'll need

$$f'(z) \rightarrow 1 \quad \text{as} \quad |Z| \rightarrow \infty. \quad (4.45)$$

The second thing is about the conformal nature of the map. Normally, local angles are preserved, as we saw in Example 4.5. However, this is only true if the mapping function $f(z)$ has a non-vanishing first derivative everywhere. At points where the first derivative is zero, though, the angle changes; in general, the local angle is multiplied by n , where n is the order of the first non-vanishing derivative of $f(z)$.

4.2.3 Flow Past an Elliptical Cylinder

Let's try the following transformation:

$$Z = f(z) = z + \frac{c^2}{z}, \quad (4.46)$$

where c is a constant. This is called the *Joukowski transformation*, and was important in the development of the theory of aerodynamics. Note that the derivative is

$$f'(z) = 1 - \frac{c^2}{z^2},$$

so $f'(z) \rightarrow 1$ as $|z| \rightarrow \infty$; this mapping won't affect the flow far away from the object. Also note that the first derivative is non-zero almost everywhere, so local angles will be preserved. However, something interesting happens at $z = \pm c$ – at that point the first derivative becomes zero, and the first non-vanishing derivative is the *second*. This means at those points local angles will be *doubled*.

We'll need the inverse of the Joukowski transformation as well; it is given by

$$z = \frac{1}{2}Z \pm \sqrt{\frac{1}{4}Z^2 - c^2}. \quad (4.47)$$

Note that, except at the points $Z = \pm 2c$, this inverse is *multivalued*. In the language of complex analysis, we would say that $Z = \pm 2c$ are *branch points*, and we could cut the plane between them to ensure the inverse is single-valued. However, in practice, we'll only need the positive root; it turns out that the negative root maps points in the Z complex plane to points within a circle of radius c on the z complex plane, while the positive root maps them to points outside that circle. Since we're studying fluid

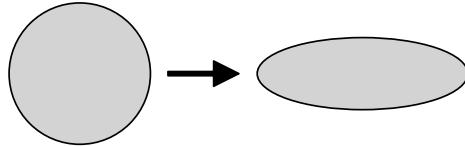


Figure 4.12: The Joukowski transformation takes a circle in the z complex place to an ellipse in the Z complex plane. Here the parameter $c = 0.7$.

flow around objects, there won't be any fluid within that radius anyway – the object itself will be there.

What does the Joukowski transform actually do? Consider its affect on a circle of radius a in the complex z plane, which is given by

$$z = ae^{i\phi}.$$

Applying the transformation leads to

$$Z = z + \frac{c^2}{z} = ae^{i\phi} + \frac{c^2}{a}e^{-i\phi}.$$

This might not look familiar, but with a bit of work we can see what it describes. Write $Z = X + iY$ and $e^{i\phi} = \cos \phi + i \sin \phi$ and separate Z into its real and imaginary components:

$$X + iY = \left(a + \frac{c^2}{a}\right) \cos \phi + i \left(a - \frac{c^2}{a}\right) \sin \phi.$$

Now take the complex conjugate of both sides and rearrange to get

$$\frac{X^2}{(a + c^2/a)^2} + \frac{Y^2}{(a - c^2/a)^2} = 1. \quad (4.48)$$

This is the equation of an ellipse with semimajor axis $a + c^2/a$ and semiminor axis $a - c^2/a$. The shape of the ellipse is controlled by the parameter c , so that as $c \rightarrow a$, the ellipse gets flatter and flatter. Figure 4.12 shows this Joukowski transformation.

To see what the flow around the ellipse looks like, we start with the flow around a circular cylinder of radius a – including the angle and circu-

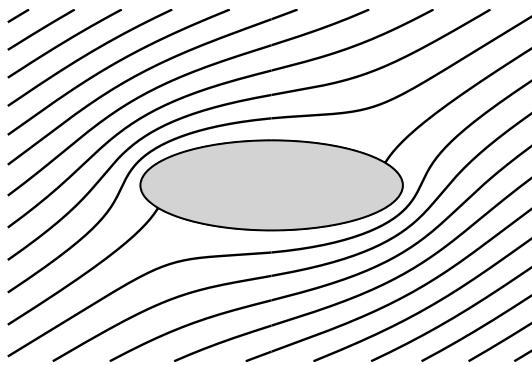


Figure 4.13: The streamlines for flow around an elliptical cylinder, with $c = 0.7$, $\alpha = \pi/6$, and $\Gamma = 0$.

lation, as given by equation (4.37) – and use the inverse Joukowski transformation (equation 4.47) to replace all the zs with Zs . After a bit of rearranging, the complex potential around an elliptical cylinder is given by

$$W(Z) = Ue^{-i\alpha} \left(\frac{1}{2}Z + \sqrt{\frac{1}{4}Z^2 - c^2} \right) + Ue^{i\alpha} \frac{a^2}{c^2} \left(\frac{1}{2}Z - \sqrt{\frac{1}{4}Z^2 - c^2} \right) - \frac{i\Gamma}{2\pi} \ln \left(\frac{1}{2}Z + \sqrt{\frac{1}{4}Z^2 - c^2} \right) \quad (4.49)$$

To find the streamlines we have to extract the imaginary part of this very complicated complex potential. Although possible, I'll leave it as an exercise for you to try; the streamlines themselves are shown in Figure 4.13. Luckily, producing the streamline plot with a computer is much easier than finding the streamlines by hand.

4.2.4 Flow Past a Flat Plate

The shape of the ellipse is dependent on the parameter c in the Joukowski transformation, and if we let $c \rightarrow a$, so that the transformation is

$$Z = z + \frac{a^2}{z}, \quad (4.50)$$

the ellipse collapses to a flat plate. Looking at equation (4.48), we see that the semiminor axis goes to zero, while the semimajor axis becomes $2a$ – so the plate has a total length of $4a$. The flow in this case is shown in the left panel of Figure 4.14.

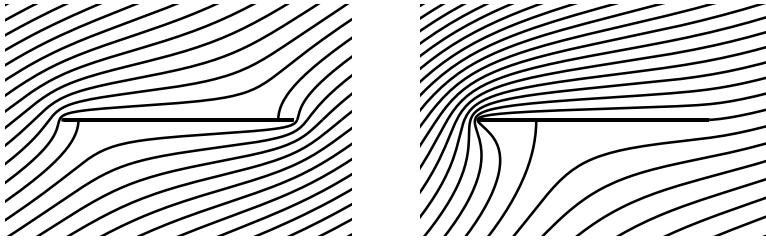


Figure 4.14: The streamlines for flow around a flat plate with angle $\alpha = \pi/6$.
Left: $\Gamma = 0$. Right: $\Gamma = -4\pi U a \sin \alpha$.

Although it's difficult to see in that streamline plot, there are two points in the flow where the velocity goes to infinity – there are two *singularities*. They occur at each edge of the plate, at $Z = \pm 2a$. To investigate the velocity of the flow, we have to find the derivative dW/dZ , which can be lengthy and complicated. To simplify things for a bit, suppose $\Gamma = 0$; letting $a = c$ in equation (4.49) and taking the derivative, we get (with some simplification)

$$\mathbb{U} - i\mathbb{V} = \frac{dW}{dZ} = U \cos \alpha - i \frac{U \sin \alpha Z}{\sqrt{Z^2 - 4a^2}}.$$

You can see in the imaginary term that the denominator goes to zero as $Z \rightarrow \pm 2a$ – so the Y component of the velocity blows up there and we have a singularity in the flow.

Now, a singularity is not physical, so presumably in a real fluid flowing past a thin flat plate they would not exist. One way to remove them is to set $\alpha = 0$, but that's not very general. We can do better, but we need to resort to a bit of a trick to analyze this situation. If we include the circulation term in equation (4.49), the derivative gets even messier, so let's use equation (4.44) to find the velocity components \mathbb{U} and \mathbb{V} :

$$\mathbb{U} - i\mathbb{V} = \frac{dW}{dZ} = \frac{dw/dz}{f'(z)} = \frac{Ue^{-i\alpha} - U(a^2/z^2)e^{i\alpha} - i\Gamma/2\pi z}{1 - a^2/z^2}. \quad (4.51)$$

Notice that the velocity goes to infinity at $z = \pm a$, which corresponds in the Z complex plane to $Z = \pm 2a$ as we just found. At this point, we should use the inverse Joukowski transformation, equation (4.47), to replace all the zs with Zs , but it's not necessary as long as we're careful.

If you look at the velocity, you might notice that we can remove *one* singularity from the flow by choosing just the right amount of circulation: if

$$\Gamma = -4\pi U a \sin \alpha, \quad (4.52)$$

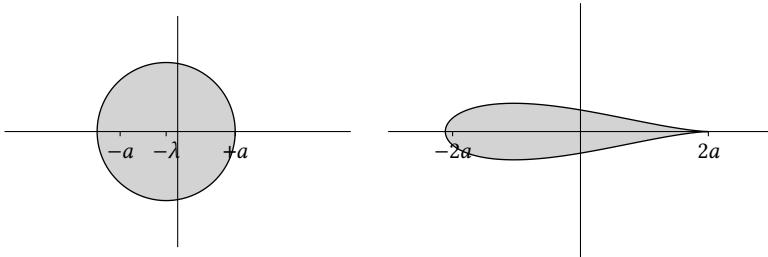


Figure 4.15: The Joukowski transformation takes the circle, shifted to the left by an amount λ , into an aerofoil shape.

then the numerator of equation (4.51) goes to zero at $z = +a$ along with the denominator. You can show that, in this case, the flow remains finite at $z = a$ and becomes (see Problem 4.13)

$$U = U \cos \alpha \quad \text{and} \quad V = 0 \quad \text{at} \quad Z = 2a. \quad (4.53)$$

The flow with this choice is shown in the right hand panel of Figure 4.14. We've removed the singularity at the right edge of the plate – called the *trailing edge* – with an astute choice of circulation, but the singularity still exists at the *leading edge*. To fix that one, we'll have to do one more clever trick.

4.2.5 Flow Past an Aerofoil

The trick is simple – we're going to hide the leading edge singularity *inside* the object, where there is no fluid. To do that, we'll shift the circle in the z complex plane to the left by an amount λ ; see the left panel of Figure 4.15 for how this looks. Note that we still want the circle to extend to $z = +a$ on the right side, so we'll also increase the radius of the circle to $a + \lambda$. Even if λ is a small value, $z = -a$ will be covered by the circle, and when transformed to the Z complex plane, the point $Z = -2a$ will be covered by the object the circle is transformed into under the Joukowski transformation – a classic aerofoil shape, as shown in the right panel of Figure 4.15.

The complex potential around the shifted circle can be found by replacing z with $z + \lambda$ and a with $a + \lambda$, and it becomes

$$w(z) = U \left[(z + \lambda) e^{-i\alpha} + \frac{(a + \lambda)^2}{z + \lambda} e^{i\alpha} \right] - \frac{i\Gamma}{2\pi} \ln(z + \lambda). \quad (4.54)$$

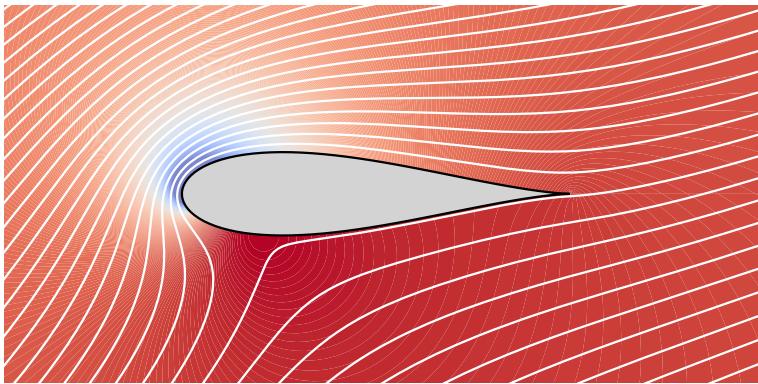


Figure 4.16: The streamline and pressure of the flow around an aerofoil. Here $\alpha = \pi/6$, $\Gamma = -4\pi U(a + \lambda) \sin \alpha$, and $\lambda = 0.2$.

As usual, to find the complex potential $W(Z)$ around the aerofoil, we replace z with Z using the inverse Joukowski transform, equation (4.47). Feel free to try that on your own. But we can also examine the velocity of the flow around the aerofoil with the same trick we did above – using equation (4.44):

$$\mathbb{U} - i\mathbb{V} = \frac{dw/dz}{f'(z)} = \frac{Ue^{-i\alpha} - Ue^{i\alpha}(a + \lambda)^2/(z + \lambda)^2 - i\Gamma/2\pi(z + \lambda)}{1 - a^2/z^2} \quad (4.55)$$

From this equation, it looks like we still have singularities at $z = \pm a$ or $Z = \pm 2a$. However, if we once again specify the circulation, we can remove the singularity at $z = +a$; we need to set

$$\Gamma = -4\pi U(a + \lambda) \sin \alpha \quad (4.56)$$

for the flow to remain finite there. If the aerofoil is “thin,” so that $\lambda \ll a$, then Γ reduces to that for the flat plate. This value of the circulation is called the *Kutta-Joukowski condition*, something we’ll discuss in more depth momentarily.

What about the singularity at $z = -a$ or $Z = -2a$? We don’t have to worry about that one – it’s covered by the aerofoil, and so there is no fluid there. Our trick worked, and as long as we have the right circulation, the flow is everywhere finite around the wing. The streamlines are shown in Figure 4.16.

Now that we have the velocity $[\mathbb{U}, \mathbb{V}]$ of the fluid around the aerofoil, we could in principle find the pressure from Bernoulli’s principle – we’re

still dealing with steady, irrotational flow, so H is constant within the fluid, and we can write

$$\frac{p_\infty}{\rho} + \frac{1}{2}U^2 = \frac{p(s, \phi)}{\rho} + \frac{1}{2}\mathbf{u}^2,$$

calculate $\mathbf{u}^2 = U^2 + V^2$ and solve for the pressure $p(s, \phi)$. As usual, that's a complicated procedure by hand, but easier with a computer, and Figure 4.16 shows the pressure around the aerofoil.

Then, given the pressure, we could integrate around the aerofoil to find the force – again, too complicated for us to do here. But it *is* possible to do using more advanced complex analysis techniques, and it turns out that, regardless of the shape of the object, the result is always the same:

$$\mathbf{F} = -\rho U \Gamma \hat{\mathbf{y}}, \quad (4.57)$$

where the coordinate system is chosen such that the uniform flow far away is directed along the positive x axis. This is the same result as for a circular cylinder, equation (4.38), and it's correct for the aerofoil as well. This general result is called the *Kutta-Joukowski theorem*.

That brings us finally to a classic fluid dynamics question: how do airplanes fly? The answer, of course, as anyone who has stuck their hand out the window of a moving car knows, is that the wing deflects the air downward, causing an upward force on the wing itself. However, the Kutta-Joukowski condition and theorem give an alternative explanation.

We need two bits of information. First, recall Kelvin's Circulation Theorem from Section 3.1.4, which says that the circulation Γ around a dyed, closed curve $C(t)$ in the fluid is constant. And second, we know that for there to be a singularity-free flow around an aerofoil, there must be circulation $\Gamma = -4\pi U a \sin \alpha$, and that this circulation is what provides upward lift on the aerofoil. Consider, then, what happens as an airplane starts moving down the runway to take off – see Figure 4.17.

When the airplane is initially at rest, before starting takeoff (top panel), the circulation around the curve $C(t)$ is obviously *zero* – the wing and fluid are at rest. Then the wing starts to move to the left, or, as in our reference frames above where the object is at rest, the fluid starts to move to the right. For there to be no singularities in the flow, there must be circulation around the wing – but the *total* circulation is still zero from Kelvin's Circulation Theorem. So there must be some *additional* circulation within $C(t)$, and in the opposite sense to balance out. This is called a *starting vortex*, and is easily seen in real airplanes or even with some water, food colouring, and a plastic card, as shown in Figure 4.18. Notice the vortex is counterclockwise, as expected. Why does the airplane fly?

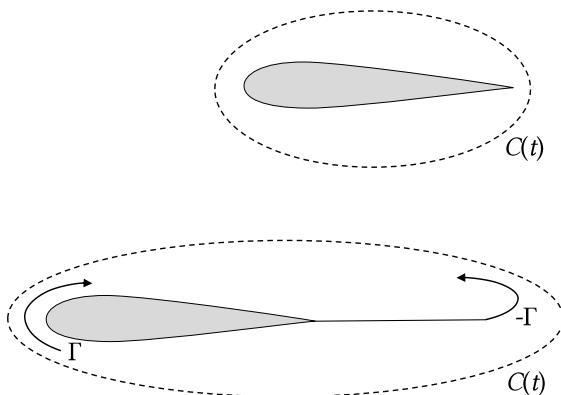


Figure 4.17: An airplane taking off induces a starting vortex.

Because the circulation generated around the wing provides *lift*. As the airplane travels faster (so larger U), both the circulation and the lift force increase.

4.3 Vortex Motion

4.3.1 The Helmholtz Vortex Theorems

We've investigated flows involving vorticity throughout the book, and the time has come to properly define a *vortex line* – not quite the same thing as a line vortex flow – and a *vortex tube*.

A vortex line is a curve in the fluid which, at some time t , has the same direction as the vorticity,

$$\boldsymbol{\omega} = \nabla \times \mathbf{u}$$

at each point (see left panel of Figure 4.19). Essentially that makes vortex lines similar to stream lines, but for vorticity $\boldsymbol{\omega}$ rather than the velocity \mathbf{u} . A vortex tube is a bundle of vortex lines – it's formed, again at some time t , by all vortex lines passing through a closed curve C_1 (see the right panel of Figure 4.19).

For an ideal fluid, Helmholtz formulated a number of theorems³ about the appearance and motion of vortex lines and tubes:

³I've seen textbooks list two, three, or four theorems; I'll state three here, but the essential ideas are all the same.

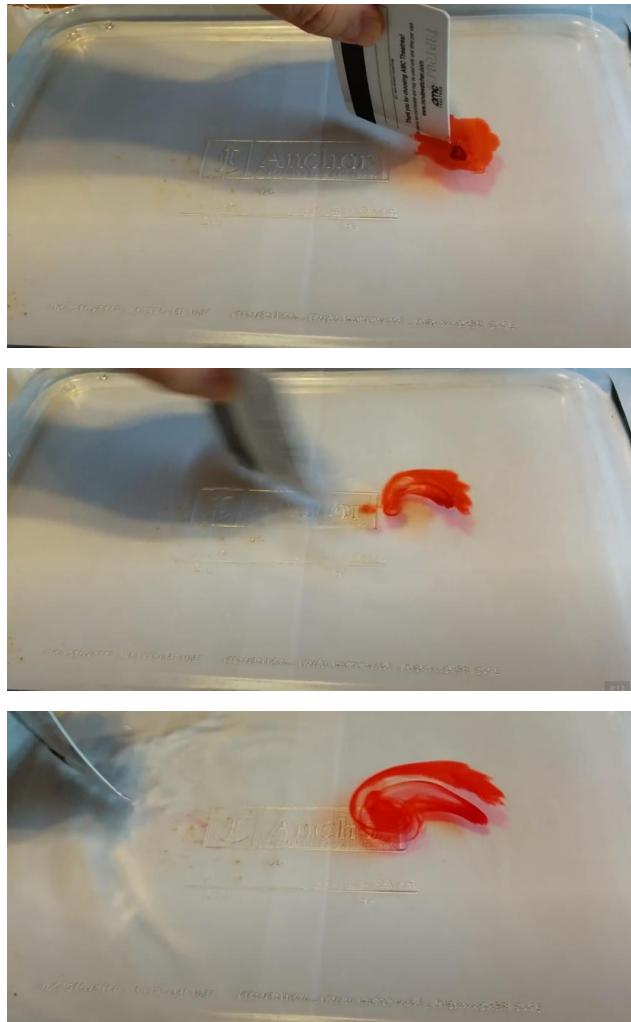


Figure 4.18: Put some water in a shallow container, place a small plastic card in the fluid, and put a small drop of food colouring at the edge of the card. Drag the card to the left, keeping the angle of attack consistent, and the dye will trace out the starting vortex.

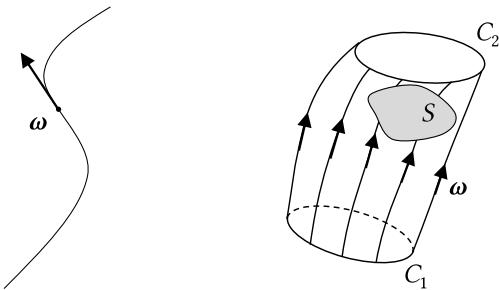


Figure 4.19: Left: The tangent of each point on a vortex line has the same direction as the vorticity ω . Right: A vortex tube is the collection of vortex lines that lie along a closed curve in the fluid.

Helmholtz's Vortex Theorems.

1. Vortex lines move with the fluid.
2. The strength of a vortex tube is constant.
3. A vortex tube cannot end within the fluid – it must end on a solid boundary, a free surface, or form a closed loop.

Of these, the first is most important to our purpose here, and I'll sketch a brief proof. Consider the surface S on the vortex tube as shown in Figure 4.19 – S is on the surface of the vortex tube itself, so that its area vector \hat{n} is perpendicular to the vorticity ω of each vortex line. If that's the case, then the circulation around the perimeter of S must be zero. We can see this from equation (3.16), which says the circulation can be calculated from

$$\Gamma = \int_S \omega \cdot d\mathbf{a}.$$

Since the area vector is always perpendicular to the vorticity, the dot product is zero and $\Gamma = 0$ around this surface. Kelvin's circulation theorem ensures that $\Gamma = 0$ always for the surface S , so that even as the fluid moves, S remains on the surface of the vortex tube; since S is an arbitrary surface and could be anywhere, the conclusion is that as the fluid moves, the vortex tube remains a vortex tube and moves with the fluid. Taking

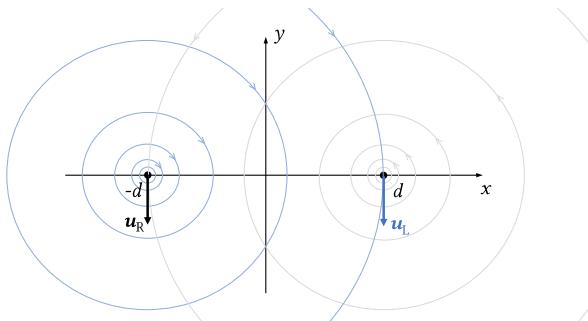


Figure 4.20: Two vortexes of equal but opposite circulation, a distance $2d$ apart. The left vortex induces a fluid velocity \mathbf{u}_L at the position of the right vortex, and the right vortex induces a velocity \mathbf{u}_R at the left.

the vortex tube to be infinitesimally thin gives us a vortex line, proving Helmholtz's first theorem.

4.3.2 A Vortex Pair

The practical result of this theorem can be seen by considering two vortexes, separated by a distance $2d$ and having opposite circulation but equal strength; this is the same flow we've seen before for a vortex beside a wall (Section 4.1.3), but here we'll actually consider both vortexes in the fluid. Helmholtz's first theorem says that each vortex will move with the fluid – in this case, the vortex on the left will have an effect on the one on the right, and vice versa. To see what effect they have on each other, take a look at Figure 4.20 and consider first the left vortex. The fluid velocity due to it is given by

$$\mathbf{u} = \frac{\Gamma}{2\pi s} \hat{\phi},$$

which at the position of the right vortex is

$$\mathbf{u} = -\frac{\Gamma}{4\pi d} \hat{\mathbf{y}}.$$

Evidently, the left vortex causes the right vortex to move *down* along the negative y direction. Note that the right vortex has no effect on itself – it can't push on itself any more than a charged particle can create an electric force on itself.

If we look at the effect of the right vortex on the left, we get a similar result: the right causes the left to move downward with the same speed.

Both vortexes move together, in lockstep, with speed

$$V = \frac{\Gamma}{4\pi d}. \quad (4.58)$$

It's actually quite easy to create a vortex pair like this; dragging a small plate through water will do it, with a vortex formed at each edge of the plate. Once the plate is removed, the vortexes will continue to move due to each other's influence. Figure 4.21 shows a vortex pair moving across a small pool.

If we want to model the vortex pair, we can shift to a reference frame in which they are at rest so that the overall flow is steady. That means we'll need to include, in addition to the two vortexes, a uniform flow with speed V moving upward. The complex potential of uniform flow is

$$w_{\text{uni}}(z) = Uze^{-i\alpha},$$

and to have the flow move upward along $+\hat{y}$, we'll set $\alpha = \pi/2$, so that $e^{-i\pi/2} = -i$. With $U = V = \Gamma/4\pi d$ we have

$$w_{\text{uni}}(z) = -\frac{i\Gamma z}{4\pi d}.$$

The full complex potential, including the two opposite but equal vortexes, would be

$$w(z) = -\frac{i\Gamma z}{4\pi d} - \frac{i\Gamma}{2\pi} \ln(z-d) + \frac{i\Gamma}{2\pi} \ln(z+d), \quad (4.59)$$

and streamlines for this flow are shown in Figure 4.22.

Are the vortexes actually at rest in this reference frame? We can check easily enough by calculating their speed. Let's start with the derivative of the complex potential,

$$\frac{dw}{dz} = u - iv = -\frac{i\Gamma}{4\pi d} - \frac{i\Gamma}{2\pi} \frac{1}{z-d} + \frac{i\Gamma}{2\pi} \frac{1}{z+d}. \quad (4.60)$$

To find the speed of the vortex on the right, we evaluate this derivative at its location, $z = d$:

$$u_R - iv_R = \left. \frac{dw}{dz} \right|_{z=d}.$$

But be careful here: the right vortex won't be affected by itself, so we'll ignore that particular term –the second – in equation (4.60). The speed of the right vortex becomes

$$u_R - iv_R = -\frac{i\Gamma}{4\pi d} + \frac{i\Gamma}{2\pi} \left(\frac{1}{2d} \right) = 0.$$

So, as expected, $u_R = 0$ and $v_R = 0$ – the vortex on the right is at rest in this reference frame. Similarly for the vortex on the left.

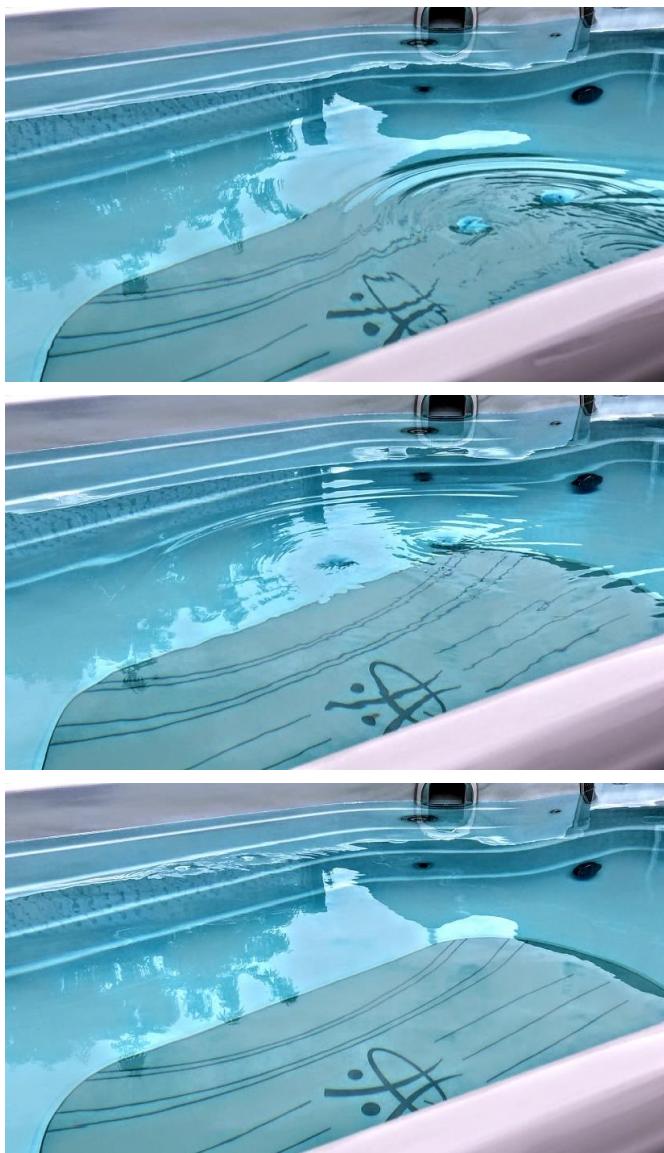


Figure 4.21: It's possible to create a vortex pair by dragging a small plate through a pool of water. The starting vortexes that form are equal in strength and opposite in direction, and they travel across the pool at constant speed and without losing strength.

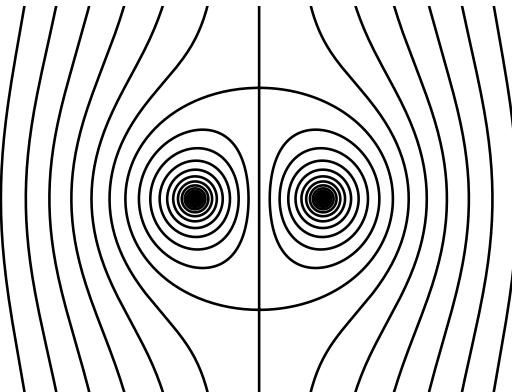


Figure 4.22: Streamlines for the flow in a reference frame in which the vortex pair is at rest.

4.3.3 Vortex Shedding

We've been discussing ideal fluids exclusively for the last two chapters, and will continue with them for the next one as well. But I want to return for a moment to the major difference between viscous and inviscid flow at high Reynolds number: the presence of a boundary layer. In an ideal fluid, flow around a circular cylinder looks like our solution (Figures 4.9 and 4.10) regardless of how fast the fluid is flowing. In reality, for high speed flow (or equivalently high Reynolds number flow), the viscous boundary layer plays an important role in the behaviour of the fluid downstream.

To illustrate this, Figure 4.23 shows a computational fluid dynamics simulation of viscous flow past a circular cylinder. At low Reynolds number (between five to 40 or so), the boundary layer at the trailing edge of the surface grows; this boundary layer is high in vorticity and eventually two small vortices appear. These vortices are equal in strength and opposite in direction – the flow *reverses* near the trailing stagnation point. Figure 4.23a shows the vortices for $R = 10$; at slightly higher Reynolds number ($R = 30$) these vortices are larger as in Figure 4.23b. However, at Reynolds number greater than around 40, something interesting happens: the vortices become *unstable* and one grows in size over the other (Figure 4.23c). The larger vortex then detaches from the cylinder and travels downstream (Figure 4.23d), forming a wake. This process is called *vortex shedding*, and the wake has a highly structured form, something we'll consider in depth in a moment.

Before we consider the wake, however, I'd like to try modelling the

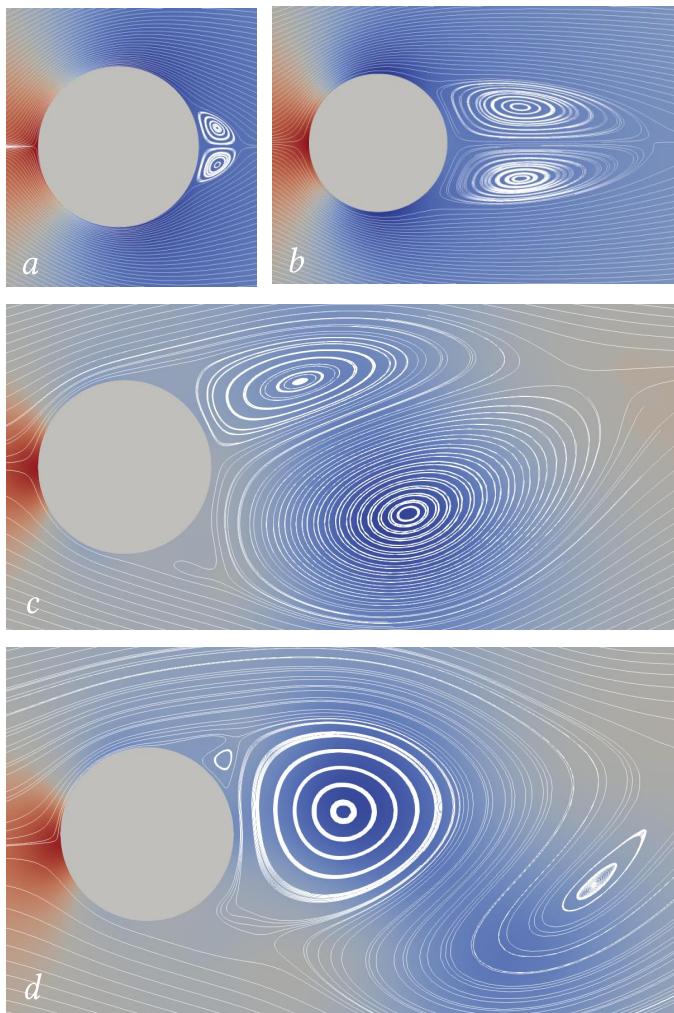


Figure 4.23: Computational fluid dynamics simulation past a circular cylinder. The popular open source code SU2 was used. (a) Reynolds number $R = 10$. A vortex pair is formed on the trailing side shortly after the flow is started. (b) $R = 30$. The region in which the fluid reverses is larger. (c) and (d) $R = 10,000$. The flow becomes unsteady and asymmetric (c), and eventually (d) the vortices begin to detach and are shed downstream.

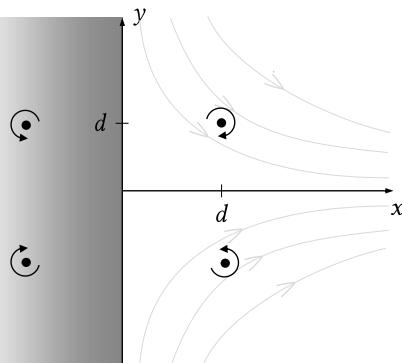


Figure 4.24: Modelling the attached vortexes requires mirror images for each. Note the circulation directions and positions of each vortex.

initial phase of vortex shedding – the steady vortexes that are attached to the trailing edge of the cylinder. We can approximate these attached vortexes as ideal line vortexes, but it turns out that adding them to a circular cylinder is a little difficult. Instead, we can simplify the situation a bit by using a flat boundary at $x = 0$, as shown in Figure 4.24. Now, that's definitely not a curved circular surface, but as long as we imagine the circle is large and the vortexes close to it, it doesn't do too bad a job.

We can model the flow around the “circle” with the stagnation point flow,

$$w(z) = \frac{1}{2}\alpha z^2,$$

as shown in the light grey lines in Figure 4.24. The vortexes will be added to this flow, but we have to be careful. There's a boundary at $x = 0$ and we must ensure the flow meets the boundary conditions there – that is, it has to slip along that wall. But we already know how to handle this – with method of images. So in addition to the two vortexes we want in our flow, we also need two mirror vortexes on the other side of the wall with opposite circulation; that will give us the correct boundary, as you can check for yourself. So, we need four vortexes, all the same strength, but with two having counterclockwise circulation (positive Γ) and two having clockwise (negative Γ). Setting the position z_0 of the top right vortex fixes all the others in place. If

$$z_0 = d + id,$$

then the lower right vortex is at $d - id = z_0^*$, the upper left is at $-d + id = -z_0^*$, and the lower left vortex is at $-d - id = -z_0$. The full complex

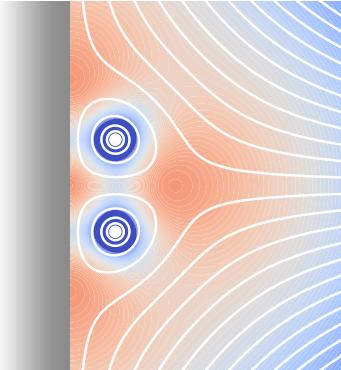


Figure 4.25: Streamlines and pressure field for the attached vortexes.

potential, including the background stagnation point flow, will be

$$w(z) = \frac{1}{2} \alpha z^2 + \frac{i\Gamma}{2\pi} \ln(z - z_0) - \frac{i\Gamma}{2\pi} \ln(z - z_0^*) - \frac{i\Gamma}{2\pi} \ln(z + z_0^*) + \frac{i\Gamma}{2\pi} \ln(z + z_0). \quad (4.61)$$

Only for one particular choice of d , however, will the vortexes be at rest – and so “attached” to the cylinder. Problem 4.14 asks you to find this; the answer is

$$d = \sqrt{\frac{\Gamma}{8\pi\alpha}}. \quad (4.62)$$

Once we have the correct complex potential, we can find the streamlines from $\phi = \text{Im}(w)$ as usual, and, if we want, the pressure from Bernoulli’s theorem. Figure 4.25 shows the vortexes and pressure; it’s up to you to compare with Figure 4.23 and see if our model does a reasonable job at reproducing it. Note, however, that although the vortexes are *steady* when placed at $d \pm id$, they’re also *unstable* and would detach with any kind of perturbation.

4.3.4 The von Kármán Vortex Street

Under the right conditions, the wake created by vortex shedding from a cylinder or other object can be highly structured and periodic – the vortexes form a *von Kármán vortex street*, in which vortexes travel downstream from the object in two separate lines. This is frequently seen in nature when wind blows past an island mountain in the ocean and surrounding clouds form the vortex street; examples are shown in Figure 4.26. It’s also readily seen in fluid simulations such as the one in Figure

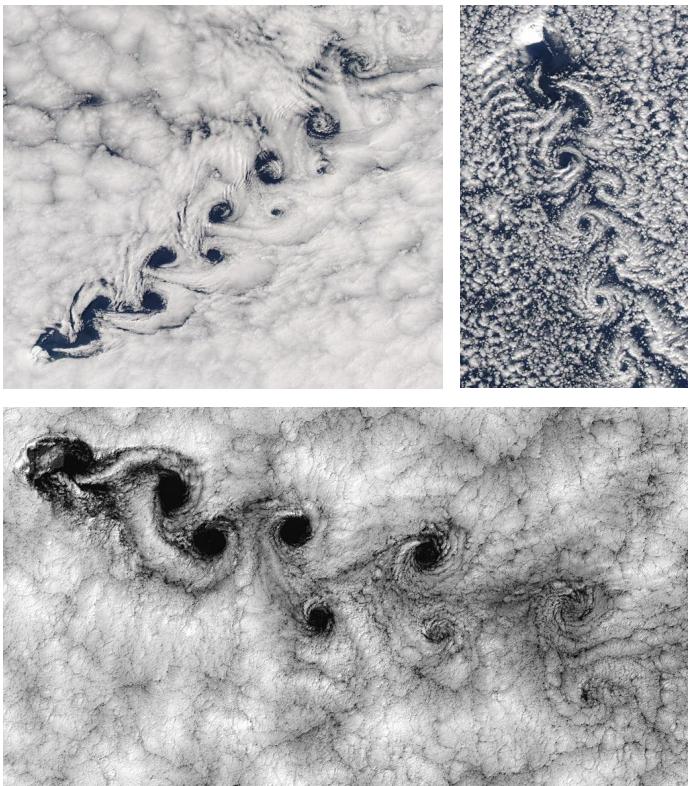


Figure 4.26: Examples of vortex streets forming in clouds as the air moves past islands in the ocean. All photos from NASA.

4.23, and, since it is at least approximately steady state, it's amenable to a simple model.

As we did above, we'll model each vortex in the street as a line vortex, but since we're approximating this as a steady state, we'll need an infinite number of them in two rows. Imagine that an object exists far to the left, and uniform flow to the right past this object creates a vortex street. We'll stagger the vortices as shown in Figure 4.27. Note the orientation of each vortex – along the top row, they have clockwise (negative Γ) circulation, while each vortex in the bottom row circulates in a counterclockwise (positive Γ) fashion. Each vortex is separated by their neighbours in their line by a distance a , the the two lines are separated by a distance b .

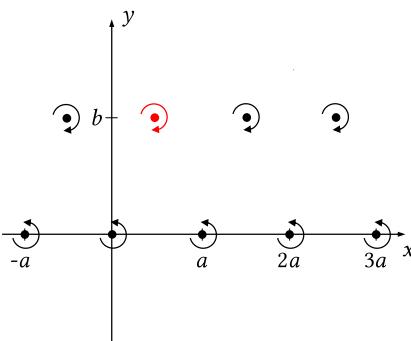


Figure 4.27: Model of a vortex street: two infinite lines of vortices, with each line having opposite rotation.

Our objective will be to find the complex potential that describes the flow, but first we need to think about how this vortex street *moves*. Consider just one vortex – the red one in Figure 4.27, which is located in the complex plane at

$$z = \frac{1}{2}a + ib.$$

Remember, each of the vortices in the street will move due to the influence of all the rest. But for this vortex, the others in the top row will have no effect – because they’re evenly spaced, they cancel in pairs, and the net result is that the fluid is at rest at the location of the red vortex. The bottom row, however, does have an effect. With a little bit of thought, you should be able to see that the fluid velocity at the position of the red vortex due to the bottom row is to the *left*.

That the flow is to the left might sound strange at first – we set up this problem so that the object creating the vortex street is far to the left and the flow is moving to the right. But right now we’re in the reference frame of the uniform flow itself – we’re moving to the right with it, and the vortices in the street evidently move to the left in this frame. That means that in the reference frame where the object is at rest, the vortices will move to the *right*, but more slowly than the uniform flow past the object. We’ll stick with the uniform flow reference frame for now, but can always add in a uniform flow later if we want to.

What exactly is the speed the vortices move with? To answer, we’ll have to write down the complex potential for the bottom row – I’ll ignore the top row for now since it has no effect on the red vortex we’re examining. In principle the complex potential is simple to write down, although

there are an infinite number of vortexes:

$$w(z) = \dots - \frac{i\Gamma}{2\pi} \ln(z+a) - \frac{i\Gamma}{2\pi} \ln(z) - \frac{i\Gamma}{2\pi} \ln(z-a) - \dots$$

Now, the n th term is

$$w_n(z) = -\frac{i\Gamma}{2\pi} \ln(z-na),$$

and n will run from $-\infty$ to ∞ . But we can write this instead as

$$w_n(z) = -\frac{i\Gamma}{2\pi} \ln\left(1 - \frac{z}{na}\right);$$

it differs from the above only by a constant, which isn't important for the complex potential. Then the full complex potential due to the bottom row is

$$w(z) = -\frac{i\Gamma}{2\pi} \sum_{n=-\infty}^{-1} \ln\left(1 - \frac{z}{na}\right) - \frac{i\Gamma}{2\pi} \ln z - \frac{i\Gamma}{2\pi} \sum_{n=1}^{\infty} \ln\left(1 - \frac{z}{na}\right). \quad (4.63)$$

The first term handles all the vortexes to the left of the origin, the middle term the vortex at the origin itself, and the right term the vortexes to the right.

This is correct, but we can simplify it quite a bit by doing some rearranging. I'll write out some of the terms so we can see the pattern we're looking for better:

$$\begin{aligned} w(z) = & -\frac{i\Gamma}{2\pi} \left[\ln z + \dots + \ln\left(1 - \frac{z}{(-2a)}\right) + \ln\left(1 - \frac{z}{(-a)}\right) + \right. \\ & \left. \ln\left(1 - \frac{z}{a}\right) + \ln\left(1 - \frac{z}{2a}\right) + \dots \right] \end{aligned}$$

We can use log rules to turn those additions of logs to multiplications within, getting

$$w(z) = -\frac{i\Gamma}{2\pi} \ln \left[(z) \dots \left(1 - \frac{z}{(-2a)}\right) \left(1 - \frac{z}{(-a)}\right) \left(1 - \frac{z}{(a)}\right) \left(1 - \frac{z}{(2a)}\right) \dots \right]$$

At this point we can do some of the multiplication. Consider the inner two terms - if we multiply them together we get a difference of squares,

$$\left(1 - \frac{z}{(-a)}\right) \left(1 - \frac{z}{(a)}\right) = \left(1 - \frac{z^2}{a^2}\right),$$

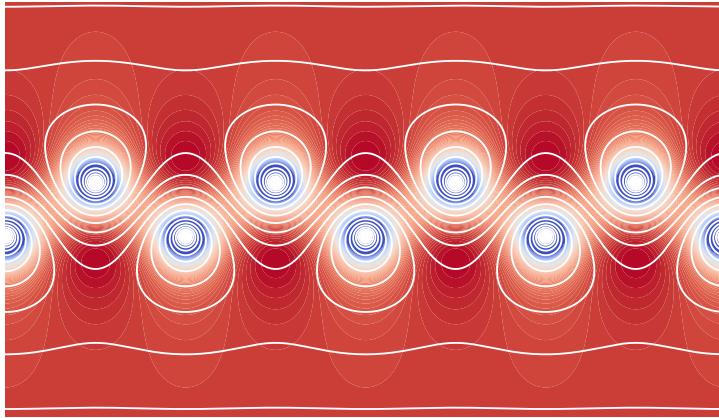


Figure 4.28: Streamlines and pressure of the simple model for the vortex street. Here $a = 1$ and $b = 0.2$ and the uniform flow velocity is $U = u_0$ so that the vortexes are at rest in this frame.

and similarly with the outer two terms. In fact, along the entire infinite number of terms, we can combine them in pairs so that we get

$$w(z) = -\frac{i\Gamma}{2\pi} \ln \left[z \prod_{n=1}^{\infty} \left(1 - \frac{z^2}{n^2 a^2} \right) \right].$$

It turns out the thing in the square brackets is actually an infinite product expansion of the sine function, and we finally have

$$w(z) = -\frac{i\Gamma}{2\pi} \ln \left[\sin \left(\frac{\pi z}{a} \right) \right]. \quad (4.64)$$

Now we can easily get the speed of the red vortex in Figure 4.27. Taking the derivative gives

$$u - iv = \frac{dw}{dz} = -\frac{i\Gamma}{2\pi} \cot \left(\frac{\pi z}{a} \right).$$

This is the velocity of the fluid everywhere due to the bottom row only. If we evaluate this at the location of the red vortex, $z = \frac{1}{2}a + ib$, we find

$$u_0 - iv_0 = \left. \frac{dw}{dz} \right|_{z=\frac{1}{2}a+ib} = -\frac{\Gamma}{2a} \tanh \left(\frac{\pi b}{a} \right).$$

That means the velocity components of the vortex are

$$u_0 = -\frac{\Gamma}{2a} \tanh \left(\frac{\pi b}{a} \right) \quad \text{and} \quad v_0 = 0. \quad (4.65)$$

As expected, it moves to the left (u_0 is negative) and has no motion along the y direction (see Problem 4.15).

Now that we've analyzed the bottom row, we can easily repeat for the top to find the total complex potential. The full complex potential everywhere is

$$w(z) = -\frac{i\Gamma}{2\pi} \ln \left[\sin \left(\frac{\pi z}{a} \right) \right] + \frac{i\Gamma}{2\pi} \ln \left[\sin \left(\frac{\pi(z - a/2 - ib)}{a} \right) \right] + Uz, \quad (4.66)$$

where the last term allows for a uniform flow if we want to move to a reference frame in which it has a non-zero velocity. Figure 4.28 shows the streamlines and pressure for this complex potential, and I set $U = -u_0$, the speed at which the vortices travel, so that they are at rest in the figure. Again, it's up to you to decide how good the model is; compare with the cloud photographs or simulation images.

Problems

Problem 4.1 – Complex numbers.

- (a) Show that $|z|^2$ is real using polar coordinates.
- (b) Show that $1/i = -i$.
- (c) Show that

$$\sqrt{i} = \frac{1}{\sqrt{2}} + \frac{i}{\sqrt{2}}.$$

Problem 4.2 – A line source. Starting from the velocity potential $\varphi(s, \phi)$ and stream function $\psi(s, \phi)$, derive the complex potential for a line source of strength Q .

Problem 4.3 – Power law complex potentials. Consider a general complex potential of the form $w(z) = Az^n$. For $n = 2/3, 3/2, 2$, and $n = 3$, plot the streamlines and physically interpret the flow. Is there a relationship between the exponent n and how the flow behaves?

Problem 4.4 – The doublet flow. Of special interest is the power law flow with $n = -1$: $w(z) = A/z$. For this flow, find the velocity potential and stream function as well as the velocity (use polar coordinates). Plot the streamlines.

Problem 4.5 – Combination of flows. Plot the streamlines for a flow that consists of a stagnation point of strength $\alpha = 1$ and a line vortex of strength $\Gamma = 10$ and positioned at $z = 1$.

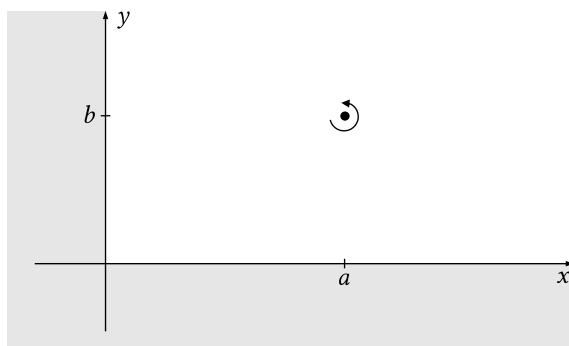


Figure 4.29: A line vortex in a corner.

Problem 4.6 – Boundary conditions from the mirror image. Starting from equation (4.23), find the velocity of the fluid $[u, v]$ and show that the flow slips along the wall (in other words, shown that $u(0, y) = 0$). Warning: this will take a fair bit of algebra.

Problem 4.7 – A vortex in a corner. Consider a vortex located at $z = a + ib$, where a and b are both positive, and there are boundaries as shown in Figure 4.29.

- (a) What is the complex potential for the fluid in the region $x \geq 0, y \geq 0$?
- (b) Plot the streamlines for the region $x \geq 0, y \geq 0$.
- (c) Show explicitly that the fluid velocity is zero at the corner.
- (d) This vortex will *move* due to the rest of the flow. Show that the path the vortex takes is given by

$$\frac{1}{x} + \frac{1}{y} = \text{constant.}$$

Problem 4.8 – A line source by a wall. Consider a line source Q a distance b from a boundary along $x = 0$ (so the fluid exists only in $x > 0$). Use the method of images to determine the complex potential, making sure to show explicitly that the boundary condition is satisfied (a drawing will do). Then find the velocity u of the flow and plot the streamlines.

Problem 4.9 – A line source by two walls. Suppose there are two boundaries, one at $y = +b$ and one at $y = -b$, with fluid in between. A line source Q sits right in the middle at $y = 0$. Show that the complex

potential is

$$w(z) = \frac{Q}{2\pi} \ln [\sinh(\pi z/2b)].$$

Problem 4.10 – Using the circle theorem. Consider the complex potential

$$w(z) = \frac{A}{2}(z^2 - 2cz)$$

where A and c are constants.

(a) Find the velocity potential and the stream function and plot the streamlines.

(b) Add a circular cylinder to the flow using the Milne-Thomson circle theorem and find the new complex potential. Show that the stream function is given by

$$\psi(x, y) = Ay \left[x \left(1 + \frac{a^2}{x^2 + y^2} \right) - c \right] \left(1 - \frac{a^2}{x^2 + y^2} \right)$$

and plot the new streamlines.

Problem 4.11 – Flow around a circular cylinder. Equation (4.37) gives the complex potential for flow around a circular cylinder, including an angle and circulation, but for this problem set $\alpha = 0$.

(a) By taking the derivative of $w(z)$, show that the velocity is given by

$$u_s(s, \phi) = U \left(1 - \frac{a^2}{s^2} \right) \cos \phi \quad \text{and} \quad u_\phi(s, \phi) = -U \left(1 + \frac{a^2}{s^2} \right) \sin \phi + \frac{\Gamma}{2\pi s}.$$

(b) Show that there are two stagnation points on the cylinder (at $s = a$) if the circulation is $\Gamma < 4\pi U a$, only one point if $\Gamma = 4\pi U a$, and none if $\Gamma > 4\pi U a$.

(c) Where is the stagnation point in the fluid if $\Gamma = 8\pi U a$? (Hint: it's not on the boundary $s = a$.)

Problem 4.12 – Force on a circular cylinder. Problem 3.4 asked you to calculate the force on a cylinder from an ideal fluid. Repeat that calculation, but this time include *circulation* Γ (you can set $\alpha = 0$, though), and show that

$$F_x = 0 \quad \text{and} \quad F_y = -\rho U \Gamma.$$

Problem 4.13 – Flow around an ellipse and a flat plate. Let's further examine the flow around the elliptical cylinder and flat plate.

(a) Starting from the complex potential for flow past a circular cylinder (equation 4.37), use the inverse Joukowski transform to derive the flow past an elliptical cylinder (equation 4.49).

(b) With $c = a$ in the Joukowski transform, the ellipse becomes a flat plate. Using equation (4.51), show that the singularity in the flow at $z = +a$ disappears if the circulation is $\Gamma = -4\pi U a \sin \alpha$, and show that the flow at that point is

$$U = U \cos \alpha \quad \text{and} \quad V = 0.$$

Problem 4.14 – Attached Vortexes. Equation 4.61 gives the complex potential for a pair of attached vortexes. By taking the derivative of this expression and evaluating it at the location of each vortex, show that the vortexes are at rest if

$$d = \sqrt{\frac{\Gamma}{8\pi\alpha}}.$$

Problem 4.15 – The vortex street. For the vortex street, the complex potential for the bottom row is given by equation (4.64). From this, show that the velocity of the red vortex in Figure 4.27 is given by equation (4.65).

Chapter 5

Waves

5.1 Gravity and Capillary Waves

5.1.1 Surface Waves

We turn our attention now to the *interface* between two fluids – in our discussion here, this will almost always be water and air. In that case the interface is called a *surface wave*. Surface waves can be generated by wind, ocean tides, surface disturbances, and underwater disturbances (an earthquake, for example, giving rise to a tsunami). Figure 5.1 shows an example of wind-generated waves on a pond.

These waves are called *gravity waves* since, as we'll see, the main force determining their behaviour is gravity. However, in some cases – typi-

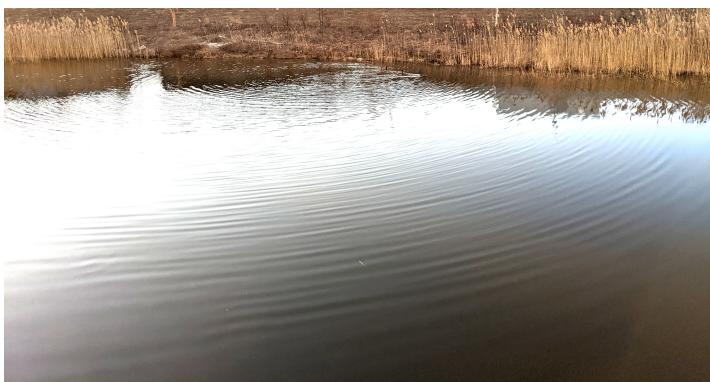


Figure 5.1: Small amplitude waves generated by wind on the surface of a small pond on the campus of Ontario Tech University.

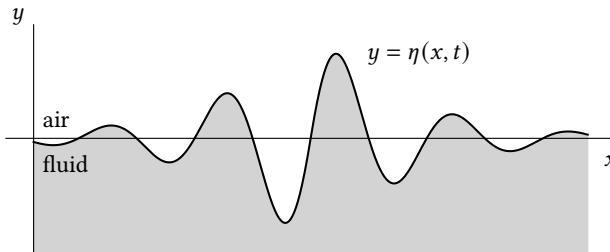


Figure 5.2: The surface of a wave is described by the function $\eta(x, t)$.

cially when the wavelengths are very short – *surface tension* can instead be the dominant force in attempting to restore equilibrium to the surface. In that case the waves can behave very differently, and often both forces must be considered.

To begin our discussion, consider a generic example of a surface wave as shown in Figure 5.2. The shape of the surface – the interface between the two fluids – will be described by the function $\eta(x, t)$, so that the surface itself is given by the equation

$$y = \eta(x, t). \quad (5.1)$$

Note that we're setting the direction of gravity to be downward along the negative y direction, in which case $\mathbf{g} = [0, -g, 0]$ in this coordinate system.

We'll assume that the fluid is two-dimensional, so there will be no z dependence or flow in that direction:

$$\mathbf{u} = [u(x, y, t), v(x, y, t), 0].$$

We'll also take the flow to be irrotational, so that $\nabla \times \mathbf{u} = 0$. In two dimensions, this means

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

It might seem like a leap to assume irrotationality, but if we imagine starting with a perfectly static fluid, with no wave at the interface, then the flow is obviously initially irrotational. But as time goes on, Kelvin's circulation theorem (see Section 3.1.4) guarantees that the flow will continue to be irrotational – the circulation must remain zero.

Since our flow is irrotational, we're free to describe it with the velocity potential $\varphi(x, y, t)$. In two dimensions, $\mathbf{u} = \nabla \varphi$ becomes

$$u = \frac{\partial \varphi}{\partial x} \quad \text{and} \quad v = \frac{\partial \varphi}{\partial y}. \quad (5.2)$$

Finally, we're still dealing with incompressible fluids, which means the velocity potential must satisfy Laplace's equation:

$$\frac{\partial^2 \varphi}{\partial x^2} + \frac{\partial^2 \varphi}{\partial y^2} = 0. \quad (5.3)$$

This lays out the guiding equations behind our discussion of waves; all that's left is to determine the boundary conditions.

5.1.2 The Kinematic and Pressure Conditions

The first boundary condition comes from the fact that any fluid particle at the surface must remain at the surface. You might be able to visualize this by imagining dyeing the fluid at the surface; as the surface moves up or down, the dyed fluid stays at the surface. This means that those fluid particles at the surface must follow the vertical motion of the wave. We can model this mathematically by defining a new function, $F(x, y, t)$, such that

$$F(x, y, t) = y - \eta(x, t). \quad (5.4)$$

Although in general F can take on any value, all fluid particles at the surface have $F = \text{constant}$ – namely, $F(x, y, t) = 0$ there since $y = \eta(x, t)$ defines our surface.

With the function F constant at the surface, we can obviously write

$$\frac{DF}{Dt} = 0 \quad \text{on} \quad y = \eta(x, t)$$

or, expanding the material derivative,

$$\frac{\partial F}{\partial t} + (\mathbf{u} \cdot \nabla)F = 0 \quad \text{on} \quad y = \eta(x, t). \quad (5.5)$$

Now,

$$\frac{\partial F}{\partial t} = \frac{\partial}{\partial t}(y - \eta(x, t)) = -\frac{\partial \eta}{\partial t}$$

and

$$(\mathbf{u} \cdot \nabla)F = u \frac{\partial F}{\partial x} + v \frac{\partial F}{\partial y} = -u \frac{\partial \eta}{\partial x} + v,$$

so equation (5.5) becomes

$$-\frac{\partial \eta}{\partial t} - u \frac{\partial \eta}{\partial x} + v = 0,$$

or

$$\boxed{\frac{\partial \eta}{\partial t} + u \frac{\partial \eta}{\partial x} = v \quad \text{on} \quad y = \eta(x, t).} \quad (5.6)$$

This is called the *kinematic condition* at the surface; it ensures that fluid particles there move with the surface wave.

The second boundary condition at the surface follows from considering the pressure in the fluid: it must be *continuous* across the interface. Since we're usually assuming air above the fluid, we'll take the pressure at the surface to be atmospheric pressure p_0 , which is of course constant along the surface.

Consider Euler's equation in the form of equation (3.6),

$$\frac{\partial \mathbf{u}}{\partial t} + (\nabla \times \mathbf{u}) \times \mathbf{u} = -\nabla \left(\frac{p}{\rho} + \frac{1}{2} \mathbf{u}^2 + \Phi \right).$$

The second term on the left is zero here, since we have irrotational flow, and we can write $\mathbf{u} = \nabla \varphi$ and then combine the velocity potential with the other terms in the gradient on the right. That gives us

$$\nabla \left(\frac{\partial \varphi}{\partial t} + \frac{p}{\rho} + \frac{1}{2} \mathbf{u}^2 + \Phi \right) = 0.$$

Integrating leads to

$$\frac{\partial \varphi}{\partial t} + \frac{p}{\rho} + \frac{1}{2} \mathbf{u}^2 + \Phi = G(t), \quad (5.7)$$

where $G(t)$ is the integration “constant” – it can still be a function of time. Of course, we're free to take $G(t)$ to be anything we want, since it will go away once we take the space derivative. So, for reasons we'll see in a second, let's take it to be

$$G(t) = \frac{p_0}{\rho}. \quad (5.8)$$

Note that it's the constant atmospheric pressure p_0 in that equation.

Our next step is to evaluate equation (5.7) at the surface $y = \eta(x, t)$:

$$\frac{\partial \varphi}{\partial t} + \frac{p_0}{\rho} + \frac{1}{2} \mathbf{u}^2 + \Phi = \frac{p_0}{\rho} \quad \text{on } y = \eta(x, t). \quad (5.9)$$

Now we see why that choice of $G(t)$ was made, since the pressure terms cancel out. Writing $\Phi = gy$ and $\mathbf{u}^2 = u^2 + v^2$, this becomes

$$\frac{\partial \varphi}{\partial t} + \frac{1}{2}(u^2 + v^2) + g\eta = 0 \quad \text{on } y = \eta(x, t).$$

(5.10)

This is the *pressure condition*, the second boundary condition that must be satisfied by the wave surface. Together with the kinematic condition and Laplace's equation, they make up the theory of surface waves. Unfortunately, although Laplace's equation is linear, the boundary conditions are not, and this makes them rather difficult to work with.

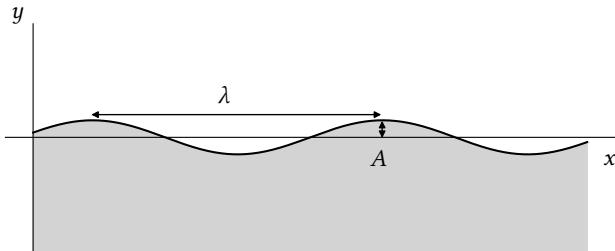


Figure 5.3: A small-amplitude wave has a wavelength $\lambda \gg A$.

5.1.3 Small Amplitude Waves

If we restrict our investigation to waves that have an amplitude that is small compared to the wavelength (see Figure 5.3), we can make some simplifications. Let's suppose that the fluid velocities u and v are small, and that therefore the surface η doesn't deviate too much from $y = 0$. If that's the case, then we can assume the derivative $\partial\eta/\partial x$ is *also* small, since the slope of the wave won't deviate too much from horizontal. If any of the terms in equations (5.6) or (5.10) have two or more of these small quantities multiplied together, we'll drop those terms – in other words, we'll drop anything that is quadratic or higher in smallness. This is called *linearizing* the boundary conditions.

Let's start with the kinematic condition, equation (5.6). The second term can be dropped since both u and $\partial\eta/\partial x$ are small. But we can do one other thing, as well. Let me write out the condition, being sure to use $y = \eta(x, t)$ in place of y :

$$v(x, \eta, t) = \frac{\partial\eta}{\partial t}.$$

Since η is close to zero, we can expand the velocity v about $y = \eta = 0$ in a Taylor expansion, and get

$$v(x, \eta, t) \approx v(x, 0, t) + \eta \frac{\partial v}{\partial y} \Big|_{\eta=0},$$

where the higher terms in the expansion were dropped. In fact, we can drop the second term, as well, since both η and v are small, and the kinematic condition becomes

$$v(x, 0, t) = \frac{\partial\eta}{\partial t} \quad \text{on } y = 0.$$

Note that we're now evaluating this at $y = 0$, since that's where the surface is to first order. One last thing: we can write this in terms of the

velocity potential rather than the velocity using $v = \partial\varphi/\partial y$ and get

$$\boxed{\frac{\partial\varphi}{\partial y} = \frac{\partial\eta}{\partial t} \quad \text{on} \quad y = 0.} \quad (5.11)$$

We can make similar arguments for the pressure condition, equation (5.10). The linearized version is

$$\boxed{\frac{\partial\varphi}{\partial t} + g\eta = 0 \quad \text{on} \quad y = 0.} \quad (5.12)$$

5.1.4 Sinusoidal Waves on Deep Water

As a simple but important example, suppose the surface can be described by a sinusoidal travelling wave, given by

$$\eta(x, t) = A \cos(kx - \omega t), \quad (5.13)$$

where $k = 2\pi/\lambda$ is the wave number and $\omega = 2\pi\nu$ is the angular frequency of the wave.

The boundary conditions will not only allow us to find the velocity potential of the fluid, but also will place important restrictions on the behaviour of the waves. The linearized kinematic condition says

$$\frac{\partial\varphi}{\partial y} = \frac{\partial\eta}{\partial t} = \omega A \sin(kx - \omega t),$$

suggesting that the potential takes the form

$$\varphi(x, y, t) \propto f(y) \sin(kx - \omega t). \quad (5.14)$$

We can find the unknown function $f(y)$ by ensuring that the potential satisfies Laplace's equation,

$$\frac{\partial^2\varphi}{\partial x^2} + \frac{\partial^2\varphi}{\partial y^2} = 0.$$

Substituting equation (5.14) into Laplace's equation leads to

$$-k^2 f(y) \sin(kx - \omega t) + f''(y) \sin(kx - \omega t) = 0.$$

Cancelling out the sine term and rearranging gives the familiar differential equation

$$f'' = k^2 f,$$

with general solution

$$f(y) = Ce^{ky} + De^{-ky}.$$

Although we've discussed at length the boundary conditions at the surface of the fluid, we haven't mentioned what happens as we go down in depth. Two options are possible: that the fluid extends to infinity below the surface (these waves are called *deep water waves*) or that a "floor" exists at some depth $y = -h$. We'll use the first possibility here, and you can explore the effects of finite depth in Problem 5.1.

If the fluid extends down to $y \rightarrow -\infty$, then we have to drop the second term of our solution by setting $D = 0$ – otherwise the potential will blow up as we go deeper into the fluid. We have, therefore,

$$\varphi(x, y, t) = Ce^{ky} \sin(kx - \omega t).$$

We can find the constant C by using the kinematic pressure condition, equation (5.11); taking the derivatives and setting $y = 0$ gives

$$kC \sin(kx - \omega t) = \omega A \sin(kx - \omega t),$$

or

$$kC = \omega A. \quad (5.15)$$

Thus, the velocity potential is

$$\varphi(x, y, t) = \frac{\omega A}{k} e^{ky} \sin(kx - \omega t). \quad (5.16)$$

This holds *everywhere* in the fluid, not just at the surface, and we can use it to explore the motion of the fluid below the waves – we'll do that in a moment. But first, what about the pressure condition? We haven't applied that yet.

Evaluating the pressure condition, equation (5.12), gives

$$-\frac{\omega^2}{k} \cos(kx - \omega t) + gA \cos(kx - \omega t) = 0,$$

or

$\boxed{\omega^2 = gk.}$

(5.17)

This is an important relationship between the frequency of a wave and its wavelength, and is called the *dispersion relation* for deep water gravity waves. You might be more familiar with *disperionless* waves like waves



Figure 5.4: A disturbance (from dropping a large stone) on the surface of a pond creates gravity waves which propagate outward. As you can see in the photograph, the largest wavelengths move fastest and thus travel furthest.

on a string or even electromagnetic waves. In these kinds of waves, $\omega \propto k$. Because sinusoidal waves travel with speed

$$c = \frac{\omega}{k} \quad (5.18)$$

(called the *phase velocity*), the speed of the wave won't depend on the wavelength. But for water waves, with $\omega \propto \sqrt{k}$, different wavelengths will travel at different speeds, since

$$c = \sqrt{\frac{g}{k}} = \sqrt{\frac{g\lambda}{2\pi}}.$$

Thus, longer wavelengths will travel *faster* than shorter ones. This leads to some interesting surface wave effects, one of which is shown in Figure 5.4 – the waves with longer wavelengths will travel further from the source of the disturbance in the pond, giving the ripples a distinctive look.

Example 5.1 – Fluid motion under the surface.

Now that we have the velocity potential $\varphi(x, y, t)$, we can examine the entire fluid flow. In particular, imagine a single fluid particle initially at some depth under the surface. What path does

the particle take as the surface waves propagate above?

Start with the fluid velocity, which we find from the potential:

$$u = \frac{\partial \varphi}{\partial x} = A\omega e^{ky} \cos(kx - \omega t),$$

and

$$v = \frac{\partial \varphi}{\partial y} = A\omega e^{ky} \sin(kx - \omega t).$$

Fluid particles, of course, follow the fluid, so we can write the change in position of a particular fluid particle as

$$\frac{dx}{dt} = u \quad \text{and} \quad \frac{dy}{dt} = v.$$

This gives us two coupled differential equations to solve. But we can uncouple them by making a reasonable assumption: that the fluid particle doesn't deviate too much from its *mean position* (\bar{x}, \bar{y}). That is,

$$x = \bar{x} + x' \quad \text{and} \quad y = \bar{y} + y',$$

where x' and y' are small.

Since \bar{x} and \bar{y} are constant, we can write

$$\frac{dx}{dt} = \frac{dx'}{dt} = u \approx A\omega e^{k\bar{y}} \cos(k\bar{x} - \omega t)$$

and

$$\frac{dy}{dt} = \frac{dy'}{dt} = v \approx A\omega e^{k\bar{y}} \sin(k\bar{x} - \omega t).$$

Note that now the right-hand-sides contain only the (constant) mean positions, meaning that we can simply integrate to find the particle path and get

$$x' = -Ae^{k\bar{y}} \sin(k\bar{x} - \omega t), \quad y' = Ae^{k\bar{y}} \cos(k\bar{x} - \omega t). \quad (5.19)$$

As time goes on and the surface wave travels horizontally, the fluid particles under the surface trace out *circles* of radius $Ae^{k\bar{y}}$. As the depth increases, the circles get smaller, as shown in Figure 5.5.

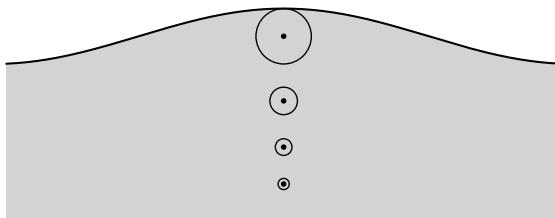


Figure 5.5: Fluid particles in deep water under surface waves move in circular paths about their mean position (the dot) with radii that get exponentially smaller with decreasing depth. If the wave is propagating to the right, the paths will be clockwise.

5.1.5 Capillary Waves

In Section 5.1.2 we took the pressure to be continuous across the fluid-air interface, so that the pressure at the surface of the fluid was atmospheric pressure p_0 . However, this isn't quite true – if the surface is *curved*, as ours of course will be, then there is a small pressure discontinuity across the surface due to the *surface tension* of the fluid. Surface tension is related to the cohesion of fluid molecules and results in a force (per unit length) that is tangential to the surface of the fluid. Water, for example, has a surface tension of $T = 72.8 \text{ mN/m}$ at a temperature of 20°C .

This tangential force will change the surface boundary conditions we just derived, although in many cases the effect can be neglected and gravity waves are a good approximation. However, for small disturbances to the surface or short wavelengths, surface tension can have noticeable and interesting effects. In cases where surface tension is the dominant force, the waves are called *capillary waves*, and where both gravity and surface tension are important the waves generated are sometimes called *gravity-capillary waves*.

We can derive the pressure discontinuity across the surface by considering a small element of the fluid ds as shown in Figure 5.6. Remember that although we're treating ds as a line element, our fluid is two-dimensional and so ds is actually an infinitesimally thin strip of area along the face of the wave. Surface tension exerts a force at either end of ds (T_1 on the left side and T_2 on the right), and it's the difference between these we want to calculate. Since we're still assuming small amplitude waves only, we can take the slope $d\eta/dx$ to be small. This has a number of consequences:

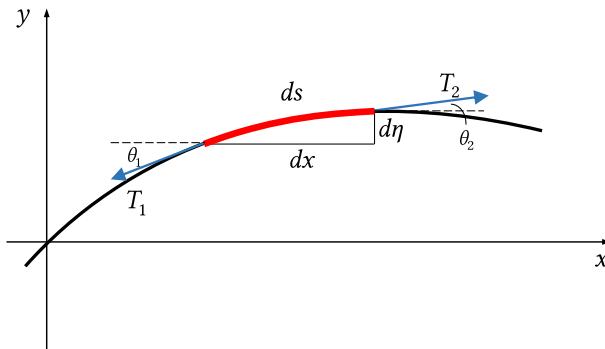


Figure 5.6: Surface tension creates a force tangential to the surface of the fluid.

- The angles θ_1 and θ_2 are small, and we can use small-angle approximations for sine and cosine, dropping any terms quadratic or higher in θ .
- The fluid element executes *vertical* motion only – it goes up and down but not side-to-side. This means that there should be no net force in the horizontal (x) direction.
- We can take $ds = \sqrt{dx^2 + d\eta^2} \approx dx$.

Examining the horizontal components of the tensions, we see that

$$T_{2x} - T_{1x} = T_2 \cos \theta_2 - T_1 \cos \theta_1 \approx T_2 - T_1.$$

But because there is no net force in this direction, we must have $T_2 = T_1$. Since the magnitudes of the two surface tensions are the same, I'll just call it T .

In the vertical direction, we have

$$T_{2y} - T_{1y} = T_2 \sin \theta_2 - T_1 \sin \theta_1 = T(\theta_2 - \theta_1),$$

but we can write the angles in terms of the slopes as

$$\theta_2 \approx \left. \frac{\partial \eta}{\partial x} \right|_{x+dx} \quad \text{and} \quad \theta_1 \approx \left. \frac{\partial \eta}{\partial x} \right|_{x-dx}$$

where x is the location of the fluid element we're considering. The difference in the vertical surface tensions is therefore

$$T_{2y} - T_{1y} = T \left(\left. \frac{\partial \eta}{\partial x} \right|_{x+dx} - \left. \frac{\partial \eta}{\partial x} \right|_{x-dx} \right) = T \frac{\partial^2 \eta}{\partial x^2} dx, \quad (5.20)$$

where the *second* derivative comes from the difference of two *first* derivatives.

Now, the quantity $T\partial^2\eta/\partial x^2$ is the net force on the fluid per unit *area* – it's a pressure. This pressure must be balanced by the difference in pressure above (atmospheric pressure p_0) and below (p) the surface, so

$$p_0 - p = T \frac{\partial^2 \eta}{\partial x^2} \quad \text{at} \quad y = \eta(x, t). \quad (5.21)$$

If the surface is concave up, then the second derivative is positive, and the pressure just above the fluid is greater than the pressure just below the surface. If the surface is concave down, the opposite happens: pressure is greater just below the surface.

We can incorporate this pressure difference into our pressure condition by repeating the steps in Section 5.1.2; when we get to equation (5.9), we need to use the new pressure at the surface,

$$p = p_0 - T \frac{\partial^2 \eta}{\partial x^2},$$

in place of just the atmospheric pressure p_0 . That just adds an extra term in the pressure condition, which becomes

$$\frac{\partial \varphi}{\partial t} + \frac{1}{2}(u^2 + v^2) + g\eta - \frac{T}{\rho} \frac{\partial^2 \eta}{\partial x^2} = 0 \quad \text{on} \quad y = \eta(x, t). \quad (5.22)$$

Linearizing as we did previously gives

$$\boxed{\frac{\partial \varphi}{\partial t} + g\eta - \frac{T}{\rho} \frac{\partial^2 \eta}{\partial x^2} = 0 \quad \text{on} \quad y = 0.} \quad (5.23)$$

With a new boundary condition, the dispersion relation we found for sinusoidal waves, equation (5.17), will be modified as well. As you can show in Problem 5.3, the new dispersion relation for gravity-capillary waves is given by

$$\boxed{\omega^2 = gk + \frac{Tk^3}{\rho}.} \quad (5.24)$$

The speed of a sinusoidal travelling wave can be found as usual from $c = \omega/k$, and is

$$\boxed{c = \sqrt{\frac{g}{k} + \frac{Tk}{\rho}}.} \quad (5.25)$$



Figure 5.7: Capillary waves created by dropping tiny pebbles into water. Notice that the shorter wavelengths have travelled further than the longer ones.

Consider the dimensionless parameter τ , defined as

$$\tau = \frac{Tk^2}{\rho g}, \quad (5.26)$$

which acts as a measure of the relative strength of the surface tension and gravity forces; notice, though, that τ also depends on wavelength through k , a point we'll return to in a moment. When $\tau \gg 1$, surface tension is the dominant force and the waves are capillary, which travel at a speed

$$c = \sqrt{\frac{Tk}{\rho}} = \sqrt{\frac{2\pi T}{\rho\lambda}}. \quad (5.27)$$

For these waves, longer wavelengths actually travel *slower*, in contrast to gravity waves where the opposite is true. This is evident in a comparison between Figure 5.4, which shows circular gravity waves generated by a disturbance, and Figure 5.7, which show circular capillary waves – much smaller in wavelength and created by a smaller disturbance.

Keep in mind, however, that both gravity and capillary waves can be present in a general disturbance depending on the wavelength of the sinusoidal wave. For water, for example, waves with a wavelength of $\lambda \sim 1.7$ cm have $\tau \sim 1$. Waves with wavelengths longer than this have $\tau \ll 1$, so gravity waves will dominate, and waves with $\lambda > 1.7$ cm will

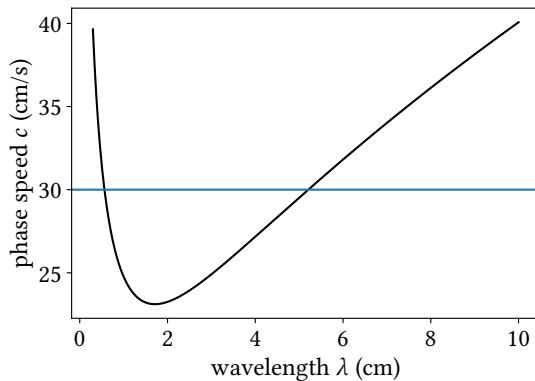


Figure 5.8: Wave speed c versus wavelength of water waves.

have $\tau \gg 1$ and capillary waves will dominate. Consider Figure 5.8, which plots the wave speed c as a function of wavelength λ (using water for values of ρ and T). There is a *minimum* speed at

$$k = \sqrt{\frac{\rho g}{T}},$$

corresponding to $\lambda = 1.7$ cm for water, which is

$$c_{\min} = \left(\frac{4gT}{\rho} \right)^{1/4}, \quad (5.28)$$

or about 23 cm/s for water. What does this mean? Consider water streaming past an obstacle with speed U ; Figure 5.9 shows a thin pole in a creek. If $U < c_{\min}$, there will be *no* steady waves generated, and if $U > c_{\min}$, there will be *two* kinds of waves generated, since two different wavelengths can produce the same wave speed, as the horizontal line in Figure 5.8 shows. For reasons that we'll see in the next section, gravity waves move downstream, but the capillary waves generated by the obstacle will actually move upstream, giving the water the unique "folding" effect seen in Figure 5.9.

5.2 Non-Sinusoidal Waves

5.2.1 Fourier Series

Although sinusoidal waves are both simple and important, there are situations where a generated wave is decidedly *not* sinusoidal. However,

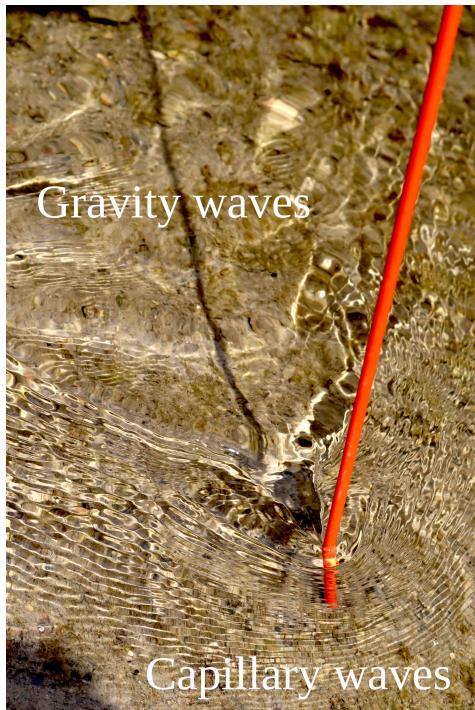


Figure 5.9: The East Oshawa Creek flows past an obstacle at a speed sufficient to produce both capillary waves (upstream) and gravity waves (downstream; they’re a little harder to see because of the much longer wavelength).

even in cases where the wave shape is complicated, the theory of Fourier analysis allows us to represent that shape as a series of sines and cosines. Perhaps a quick review of Fourier series and transforms is in order.¹

Any *periodic* function $f(x)$ can be written as a series of sines and cosines,

$$f(x) = \frac{a_0}{2} + \sum_{n=1}^{\infty} a_n \cos(nx) + \sum_{n=1}^{\infty} b_n \sin(nx),$$

or, more conveniently, as exponentials,

$$f(x) = \sum_{n=1}^{\infty} c_n e^{inx}. \quad (5.29)$$

¹For more detail on Fourier analysis, see *Mathematical Methods for Physicists*, 4th edition, by Arfken and Weber, chapters 14 and 15.

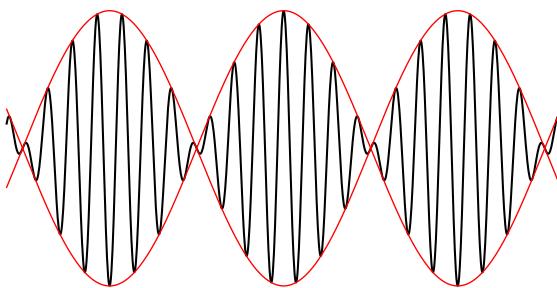


Figure 5.10: A surface wave formed by the addition of two cosine waves with similar wavenumber.

Now, for a surface wave, we'll need to include *time* as well, but that's simple enough to do; in general, we can write a travelling surface wave as

$$\eta(x, t) = \sum_{n=1}^{\infty} A_n e^{j(k_n x - \omega_n t)}, \quad (5.30)$$

where k_n and ω_n are the wave number and frequency of the n th wave.

Example 5.2 – Two cosines. Let's try a simple example and consider a travelling wave comprised of two cosines of equal amplitude,

$$\eta(x, t) = A \cos(k_1 x - \omega_1 t) + A \cos(k_2 x - \omega_2 t), \quad (5.31)$$

and further suppose that the wave numbers k_1 and k_2 are similar, so that we can write

$$k_1 = k - \Delta k \quad \text{and} \quad k_2 = k + \Delta k.$$

We'll assume the same for the frequencies ω_1 and ω_2 as well:

$$\omega_1 = \omega - \Delta\omega \quad \text{and} \quad \omega_2 = \omega + \Delta\omega.$$

If we substitute these expressions into equation (5.31), we can write the shape of the wave as (see Problem 5.5)

$$\eta(x, t) = 2A \cos(\Delta k x - \Delta\omega t) \cos(kx - \omega t). \quad (5.32)$$

This is just the equation of a single sinusoidal wave travelling with speed $c = \omega/k$, but the amplitude of this sinusoidal wave is *modulated* by the term in front, sometimes called the *envelope*. This is shown in Figure 5.10, with the envelope shown in red.

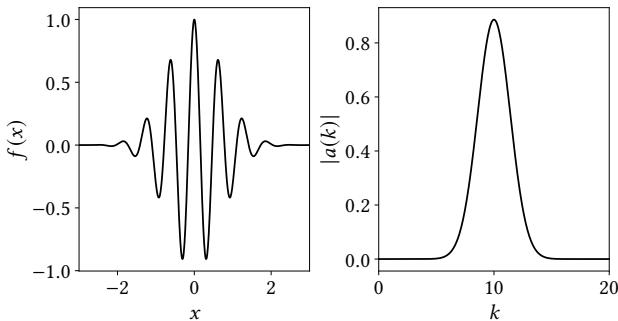


Figure 5.11: The Fourier transform of a simple wave packet. The transform $a(k)$ shows that most of the spectrum consists of wave numbers near $k_0 = 10$.

We've seen, then, that it's possible to represent more complicated wave shapes by adding together sinusoidal waves. However, a Fourier series like this will also be *periodic*, which is not always the case in nature – think of a single disturbance in a body of water generating a wave. To describe a *wave packet* like this, we need to instead use a Fourier integral.

5.2.2 Wave Packets

The Fourier integral is the generalization of the Fourier series (equation 5.29) by letting the indexed parameter n become a continuous parameter k . In that case the sum becomes an integral, and any function (including non-periodic ones) can be written as

$$f(x) = \frac{1}{\sqrt{2\pi}} \int_{-\infty}^{\infty} a(k) e^{ikx} dk. \quad (5.33)$$

Here $a(k)$ (which in general is a complex function even if $f(x)$ is real) is called the *Fourier transform* of the function $f(x)$, and can be calculated by

$$a(k) = \frac{1}{\sqrt{2\pi}} \int_{-\infty}^{\infty} f(x) e^{-ikx} dx.$$

As an example, consider the function

$$f(x) = \cos(k_0 x) e^{-x^2},$$

shown in the left panel of Figure 5.11. In the plot, I've set $k_0 = 10$. This is a classic “wave packet” shape, and is not periodic. We'll explore surface

waves that look like this in a moment; the key feature we'll use is shown in the right hand panel, which is the Fourier transform $a(k)$ (actually, it's the absolute value $|a(k)|$ since it's a complex function). Imagine the original function $f(x)$ consisting of a series of sinusoidal waves, like we did in the previous Section, except this time the index is continuous – hence the integral instead of the summation. The Fourier transform of the wave packet tells us the wave numbers of those sinusoidal waves; in this case we see that most of the waves have a wave number close to $k_0 = 10$, but that's not the only wave number involved. In fact, to make up this wave packet shape, we need a whole range of wave numbers. Despite that, finite wave packets like this do typically have a fairly narrow range of wave numbers, and we'll make use of this property in a moment.

By the way, if you've studied basic quantum mechanics, you would have seen this at play with the uncertainty principle, with wave number here standing in for momentum. The better we know the position of the wave – the less “spread out” it is – the less we know the wave number, and vice versa.

All of this suggests that a general surface wave in motion can be written as

$$\eta(x, t) = \int_{-\infty}^{\infty} a(k) e^{i(kx - \omega t)} dk \quad (5.34)$$

(compare with the Fourier series equation 5.30). Remember that surface waves will have a dispersion relation,

$$\omega = \omega(k);$$

Although we know what it is for gravity and capillary waves in deep water, in general it is an unknown function.

However, if the wave we're describing with equation (5.34) is a wave packet similar to the one in Figure 5.11, we know that the wave numbers will all be close to some central value k_0 , and we can expand the dispersion relation about that value:

$$\omega(k) \approx \omega(k_0) + (k - k_0) \frac{d\omega}{dk} \Big|_{k=k_0}.$$

In this way we can simplify the Fourier integral without knowing the exact form of the dispersion relation. First, we'll define the *group velocity* c_g by

$c_g = \frac{d\omega}{dk}$

(5.35)

(we'll see why it's called that in a moment), so that

$$\omega(k) \approx \omega_0 + (k - k_0)c_g,$$

where $\omega_0 \equiv \omega(k_0)$.

Next, we plug this expression into equation (5.34), which becomes

$$\eta(x, t) = \int_{-\infty}^{\infty} a(k) e^{i\{kx - [\omega_0 + (k - k_0)c_g]t\}} dk.$$

That exponent has gotten a little convoluted, but we can use a trick to simplify it: we'll add and subtract a term k_0x to it and then rearrange like so:

$$kx - [\omega_0 + (k - k_0)c_g]t + k_0x - k_0x = (k_0x - \omega_0t) + (k - k_0)(x - c_g t).$$

Inserting this into the integral and splitting the exponent gives

$$\eta(x, t) = \int_{-\infty}^{\infty} a(k) e^{i(k_0x - \omega_0t)} e^{i(k - k_0)(x - c_g t)} dk.$$

But the first exponential term is now *constant* – it doesn't depend on k – and can be taken outside the integral. That leaves us with, finally,

$$\eta(x, t) = e^{i(k_0x - \omega_0t)} \int_{-\infty}^{\infty} a(k) e^{i(k - k_0)(x - c_g t)} dk. \quad (5.36)$$

This might not seem like much of an accomplishment, but this equation is easy to interpret, especially if you compare with equation 5.32 from our example with adding two cosines together. Like that result, our equation here has two parts: a pure sinusoidal wave, of wave number k_0 and frequency ω_0 , and an *envelope* (the entire integral). Now, in principle this integral could be difficult or impossible to compute, depending on how complicated the initial surface disturbance was, but the important point is this: *the envelope moves with the group speed c_g* .

Example 5.3 – Deep water waves. To illustrate this, let's suppose that we have gravity waves on deep water, in which case the dispersion relation is given by equation (5.17)

$$\omega(k) = \sqrt{gk}.$$

As we've seen, this leads to a *phase velocity* (meaning the velocity of a single sinusoidal wave of frequency ω and wavenumber k) of

$$c = \sqrt{\frac{g}{k}}.$$

In a wave packet, which includes multiple sinusoidal components of different wave numbers, each component will have a different phase speed; as long as the function $a(k)$ is narrow, however, these speeds will be very similar.

The wave packet itself moves with the group speed, which in this example is

$$c_g = \frac{d\omega}{dk} = \frac{1}{2} \sqrt{\frac{g}{k}},$$

precisely half the phase speed. That means that individual sinusoidal components of the wave packet move *faster* than the wave packet. This is shown in Figure 5.12, which plots the motion of a wave packet at three different times. A single component is highlighted with a red dot; note that it moves toward the front of the wave packet as the wave packet travels to the right. This is because of the faster phase speed.

5.2.3 Group Velocity and Energy

Perhaps not surprisingly when looking at wave motion like that shown in Figure 5.12, the energy in a wave is transported at the *group velocity* c_g . What is more surprising is that this is very general – even our simple sinusoidal waves will transport energy at this speed rather than the phase speed, which is what the actual wave crests travel at.

We can show this easily enough, and while doing so explore the ideas of energy in fluids which we haven't touched on much. For example, let's start with the kinetic energy. The kinetic energy of a fluid element is of course

$$T = \frac{1}{2} m \mathbf{u}^2,$$

where $m = \rho d\tau$ is the mass of the element with infinitesimal volume $d\tau$ and $\mathbf{u}^2 = u^2 + v^2$ as usual. The kinetic energy of some volume of the fluid requires an integral over the volume to add up the contributions from each element:

$$T = \frac{1}{2} \int \rho \mathbf{u}^2 d\tau. \quad (5.37)$$

In our case, however, dealing with a two-dimensional fluid, we'll calculate the kinetic energy over an area $da = dx dy$, so that we'll have the kinetic energy per unit length in the z direction. We'll also only make use of one

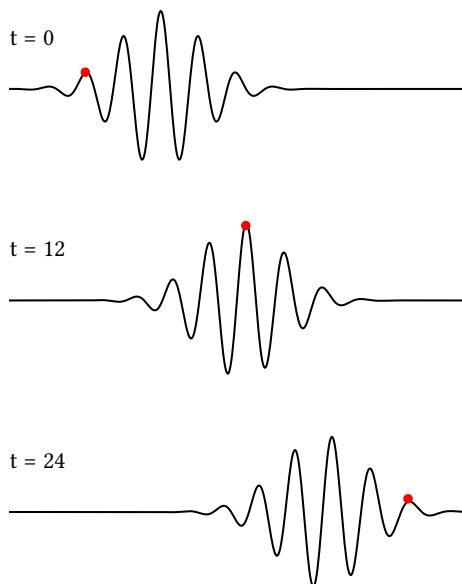


Figure 5.12: A wave packet in motion. The dispersion relation is that for deep water waves, and the individual peaks in the wave packet (one is highlighted with the red dot) move twice as fast as the wave packet itself. The time units here are arbitrary.

full wavelength as shown in Figure 5.13. The kinetic energy is then

$$T = \frac{1}{2} \rho \int_{x_0}^{x_0 + \lambda} \int_{-\infty}^{\eta} (u^2 + v^2) dy dx. \quad (5.38)$$

Assuming a sinusoidal wave with a surface given by equation (5.13) and velocity potential by equation (5.16), you can show that the result is given by (see Problem 5.6)

$$T = \frac{1}{4} \rho g A^2 \lambda. \quad (5.39)$$

This is the kinetic energy (per unit length) in the fluid below one wavelength λ of a sinusoidal surface wave.

In addition, the fluid will have gravitational potential energy as it changes in height. For a single fluid element, the potential energy is

$$U = mg y,$$

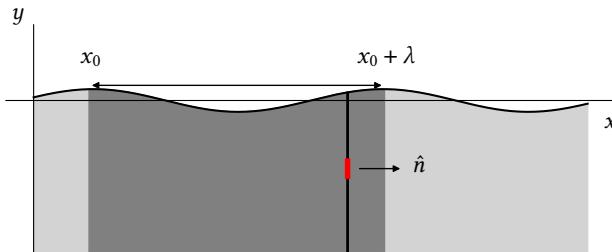


Figure 5.13: We'll calculate the energy in the fluid within one wavelength.

and integrating throughout the fluid gives

$$U = \int \rho g y d\tau. \quad (5.40)$$

As with the kinetic energy, we'll only integrate over the area shown in Figure 5.13, so the potential energy becomes

$$U = \int_{x_0}^{x_0 + \lambda} \int_0^{\eta} \rho g y dy dx. \quad (5.41)$$

Notice that the lower bound for the inner integral is $y = 0$ – we're setting the zero-point of potential energy there. For our sinusoidal waves, you can show (Problem 5.6 again) the total potential energy in one wavelength of fluid is

$$U = \frac{1}{4} \rho g A^2 \lambda. \quad (5.42)$$

This is identical to the kinetic energy, and the total energy per unit length along z is

$$E = \frac{1}{2} \rho g A^2 \lambda.$$

However, it's more customary to work with the total wave energy *per wavelength*, or

$$\bar{E} = \frac{1}{2} \rho g A^2. \quad (5.43)$$

We want to find the *rate* at which this energy flows as the wave moves. To do that, we'll calculate the *power*,

$$P = \frac{dE}{dt} = \int d\mathbf{F} \cdot \mathbf{u}.$$

Consider Figure 5.13 again. The force of the fluid on a small area $da = dydz$ (the red region) will be $d\mathbf{F} = p da \hat{x}$. Now, as above, we'll make this calculation per unit length in the z direction, so the power will be

$$P = \int_{-\infty}^{\eta} pu dy. \quad (5.44)$$

The value of the pressure in this integral is that which is a result of the waves themselves, not the hydrostatic pressure due to gravity that was discussed in Section 3.1.1. We can find that from Euler's equation in the form of (5.7); rearranging for pressure (and dropping the quadratically small velocity term), it is

$$p = -\rho \frac{\partial \varphi}{\partial t}. \quad (5.45)$$

Substituting this and $u = \partial \varphi / \partial x$ into the integral, we get

$$\frac{dE}{dt} = \frac{1}{2} \rho g \frac{\omega}{k} A^2 \cos^2(kx - \omega t).$$

If we average this expression over one full period ($2\pi/\omega$) and use the phase speed, $c = \omega/k$, it becomes

$$\frac{d\bar{E}}{dt} = \frac{1}{4} \rho g A^2 c. \quad (5.46)$$

This is the rate of energy transferred along the x direction by the wave motion; notice that it's the *average* energy. But we calculated the average energy per wavelength above, and using equation (5.43) we can write

$$\frac{d\bar{E}}{dt} = \bar{E} \left(\frac{1}{2} c \right)$$

or, since the group speed is one half the phase speed,

$$\frac{d\bar{E}}{dt} = \bar{E} c_g. \quad (5.47)$$

So, even in the case of pure sinusoidal waves, the energy is still transferred at the group speed rather than the phase speed. It turns out this is actually a general result, valid for any dispersive wave system.

5.2.4 Long-Time Response

Suppose there's some kind of disturbance in the middle of an otherwise calm pond. Although the initial response of the water will be very complicated, after enough time has passed and at a far enough distance from

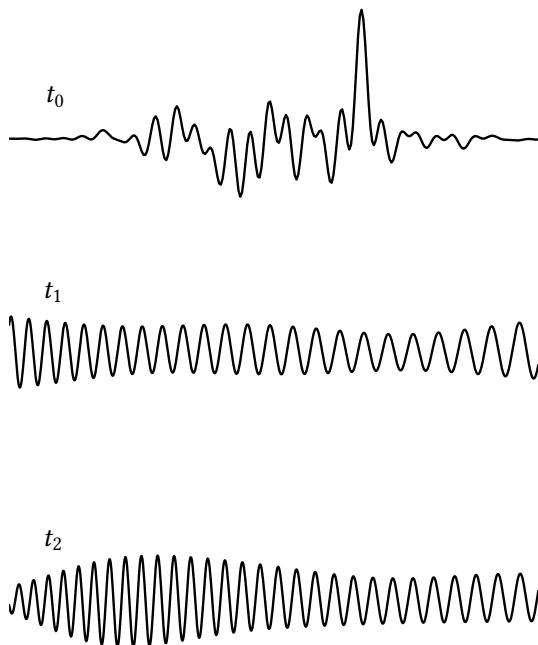


Figure 5.14: An initially messy disturbance (top panel, t_0) will eventually become regular at long times and far distances (bottom panels, t_1 and t_2).

the disturbance, the wave structure will become more regular. This is because of *dispersion* – each Fourier sinusoidal component will travel at a particular speed, $c = \sqrt{g/k}$, according to its wavelength $\lambda = 2\pi/k$, with longer wavelengths travelling faster. This is shown in Figure 5.14; the top panel, at time t_0 , displays a complicated wave packet, generated from a superposition of 20 different random frequencies. The wave packet evolves according to equation (5.34), and the middle panel, at a much later time t_1 and far from the region where the wave packet was located at t_0 , shows that the surface wave has now smoothed out and become approximately sinusoidal, but with a slowly varying wavelength, frequency, and amplitude. At time t_1 , far from the initial disturbance, the wavelength slowly increases. A short time later at t_2 , and in the exact same region, the wavelengths have become smaller.

Let's describe this long-time response to the disturbance by a *local wavenumber* $k(x, t)$ and *local frequency* $\omega(x, t)$. We can then write for the

surface the equation

$$\eta(x, t) = A(x, t)e^{i\theta(x, t)}, \quad (5.48)$$

where $A(x, t)$ describes the gradual modulation of the wave amplitude, and $\theta(x, t)$ the oscillatory aspect of it. Presumably the amplitude is complicated and depends on the initial disturbance, but we can explore the oscillatory nature of the wave by connecting the function $\theta(x, t)$ to the local wavenumber and frequency.

To do that, consider first a single sinusoidal wave; in that case we'd have

$$\theta(x, t) = kx - \omega t,$$

as usual for a travelling wave. That suggests we can write, in a more general sense,

$$k = \frac{\partial \theta}{\partial x}, \quad \text{and} \quad \omega = -\frac{\partial \theta}{\partial t}. \quad (5.49)$$

Then note that

$$\frac{\partial k}{\partial t} = \frac{\partial^2 \theta}{\partial t \partial x} = -\frac{\partial \omega}{\partial x} = \frac{\partial^2 \theta}{\partial x \partial t},$$

which means that

$$\frac{\partial k}{\partial t} + \frac{\partial \omega}{\partial x} = 0. \quad (5.50)$$

But we know that the frequency is a function of wavenumber, $\omega = \omega(k)$, so

$$\frac{\partial \omega}{\partial x} = \frac{d\omega}{dk} \frac{\partial k}{\partial x} = c_g \frac{\partial k}{\partial x},$$

where I've used the group velocity $c_g = d\omega/dk$. Equation (5.50) then becomes

$$\frac{\partial k}{\partial t} + c_g \frac{\partial k}{\partial x} = 0. \quad (5.51)$$

This is a key result, although you may not recognize it. Compare this equation to the definition of the material derivative,

$$\frac{Df}{Dt} = \frac{\partial f}{\partial t} + (\mathbf{u} \cdot \nabla) f;$$

they're the same, or would be if the velocity was purely in the x -direction. Now, if $Df/Dt = 0$, then f is constant for a particular fluid element. If we apply that line of thought to the wave result, equation (5.51), it suggests that *the local wavenumber $k(x, t)$ is constant for an observer moving with the group velocity c_g* .

Example 5.4 – Deep water waves. For deep water gravity waves, the dispersion relation is $\omega(k) = \sqrt{gk}$ and the group velocity is

$$c_g = \frac{d\omega}{dk} = \frac{1}{2} \sqrt{\frac{g}{k}}.$$

Our result above suggests that if we race along beside a wave with speed c_g , we'll always see the same wavenumber k (or wavelength λ).

Let's pretend to do exactly that – suppose a disturbance happens at $x = 0$ and $t = 0$, and we start running (somehow on top of the water!) away from it with speed c_g . After a time t has passes, we're at position

$$x = c_g t.$$

Rearranging for the speed gives

$$c_g = \frac{x}{t} = \frac{1}{2} \sqrt{\frac{g}{k}},$$

or

$$k = \frac{gt^2}{4x^2}. \quad (5.52)$$

This equation tells us what wavenumber we'll see at position x and time t .

Now, we also have

$$\omega = \sqrt{gk} = \frac{gt}{2x}. \quad (5.53)$$

Returning to our idea of a local wavenumber and frequency, we can write

$$\frac{\partial \theta}{\partial x} = k = \frac{gt^2}{4x^2} \quad \text{and} \quad \frac{\partial \theta}{\partial t} = -\omega = -\frac{gt}{2x}.$$

Integrating gives

$$\theta(x, t) = -\frac{gt^2}{4x} \quad (5.54)$$

(plus a constant which fixes the phase that I've dropped for simplicity), so the surface is given by

$$\eta(x, t) = A(x, t) e^{-igt^2/4x}. \quad (5.55)$$

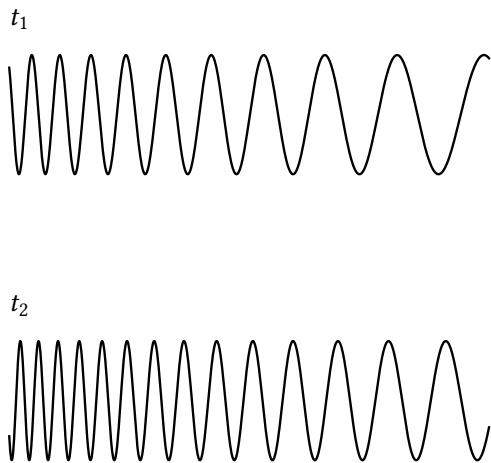


Figure 5.15: Equation (5.55) plotted at two different long times, far from the disturbance at $x = 0$.

The real part of this equation (for constant amplitude A) is plotted in Figure 5.15 for two different times. This surface matches what we saw in Figure 5.14, which plotted the motion of a complicated wave packet; in both cases the wavenumber and frequency slowly change with time and space.

5.3 Sound Waves

5.3.1 Compressible Fluids

We turn, at long last, to relaxing the one constraint on our fluids that has been in place since the very beginning – we'll allow our fluid to be *compressible*. This means that the divergence of the velocity is no longer zero; what else changes?

Well, since we're still considering inviscid fluids, Euler's equation continues to hold, which I'll write here as

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p. \quad (5.56)$$

The difference, now, though, is that the density of the fluid is no longer constant – $\rho = \rho(\mathbf{r}, t)$. We can also replace the incompressibility equation with the *continuity equation*, which expresses conservation of mass (see Problem 1.7),

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0. \quad (5.57)$$

That gives us two equations, but we actually have three unknowns: the velocity \mathbf{u} (all three components), the pressure $p(\mathbf{r}, t)$, and the density $\rho(\mathbf{r}, t)$. We need a third equation – something to connect the pressure and density. This is called an *equation of state*, and a classic example – and the one we'll assume for our fluid here – is an ideal gas.

The ideal gas law is the equation of state for an ideal or perfect gas (which of course is idealized but can model real gases under a range of conditions), and is the familiar equation

$$pV = nRT,$$

where V is the volume of the gas, n is the number of moles, and T is the temperature. It can also be written in terms of the density as

$$p = \rho \kappa T, \quad (5.58)$$

where $\kappa = R/M$ and M is the molar mass.

This gives us a third equation, linking the pressure and density, but unfortunately it also gives us another state variable: the temperature T . However, if we make another assumption – that each fluid element conserves its *entropy* – there is a simpler relation we can use. In this case, any change in the gas is *adiabatic*, meaning no heat will flow between fluid elements. This is reasonable as long as any compression or expansion of the fluid is much faster than heat can be conducted. From basic thermodynamics of ideal gases,² adiabatic processes obey the relation

$$pV^\gamma = \text{constant}, \quad (5.59)$$

where γ is the ratio of specific heats,

$$\gamma = \frac{C_p}{C_V}.$$

This ratio is $\gamma = 5/3$ for monatomic gases and $\gamma = 7/5$ for diatomic gases; air, made mostly of diatomic nitrogen and oxygen molecules, has $\gamma \approx 1.4$.

²As always, I'm giving only the bare essentials here; any basic thermodynamics textbook can fill in the gaps for you.

If we use $\rho = m/V$ and absorb the mass into the constant, we can rewrite equation (5.59) in terms of the density:

$$p\rho^{-\gamma} = \text{constant.}$$

Taking the material derivative of both sides gives

$$\frac{D}{Dt}(p\rho^{-\gamma}) = 0. \quad (5.60)$$

This is the third equation we need – a relationship between pressure and density. In words, it says that for an adiabatic process, each fluid element conserves the product $p\rho^{-\gamma}$.

5.3.2 Small Amplitude Sound Waves

Let's recap the equations that describe compressible fluids:

$$\begin{aligned} \rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) &= -\nabla p, \\ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0, \quad \text{and} \\ \frac{D}{Dt}(p\rho^{-\gamma}) &= 0. \end{aligned}$$

The simplest solution – the trivial one – to these equations is for a fluid at rest. Then $\mathbf{u} = \mathbf{0}$ and we get

$$\rho = \text{constant} = \rho_0 \quad \text{and} \quad p = \text{constant} = p_0.$$

That's obviously *too* simple, but suppose a small disturbance happens in this initially static fluid. Then the fluid won't deviate *too* much from this solution – it will have a small velocity $\mathbf{u} = \mathbf{u}_1$ and a small deviation from the constant pressure and density,

$$\rho = \rho_0 + \rho_1(\mathbf{r}, t), \quad \text{and} \quad p = p_0 + p_1(\mathbf{r}, t). \quad (5.61)$$

In these equations, the subscript “1” indicates the deviation from the static solution, and those quantities can be taken as small. As usual, that means we can drop terms that have quadratic or higher products of those quantities.

Since the fluid was initially at rest, with constant density and pressure, the product $p\rho^{-\gamma}$ initially has the value $p_0\rho_0^{-\gamma}$, which is itself constant. But that product is constant for each fluid element and can't change even after the disturbance, so the third equation above says

$$(p_0 + p_1)(\rho_0 + \rho_1)^{-\gamma} = p_0\rho_0^{-\gamma}.$$

By pulling out p_0 and ρ_0 on the left hand side, we can rewrite this as

$$\left(1 + \frac{p_1}{\rho_0}\right) \left(1 + \frac{\rho_1}{\rho_0}\right)^{-\gamma} = 1.$$

Since ρ_1 is small we can expand the second term on the left as

$$\left(1 + \frac{\rho_1}{\rho_0}\right)^{-\gamma} \approx 1 - \gamma \frac{\rho_1}{\rho_0},$$

giving us, after some multiplication,

$$1 - \gamma \frac{\rho_1}{\rho_0} + \frac{p_1}{p_0} - \gamma \frac{p_1}{p_0} \frac{\rho_1}{\rho_0} = 1.$$

But that last term on the left is quadratic in smallness and can be dropped. The ones cancel, and we're left with

$$\frac{p_1}{p_0} = \gamma \frac{\rho_1}{\rho_0}.$$

If we define the quantity

$$a_0 = \sqrt{\frac{\gamma p_0}{\rho_0}}, \quad (5.62)$$

we have that the pressure and density are proportional to each other:

$$p_1 = a_0^2 \rho_1. \quad (5.63)$$

What exactly does a_0 represent? I'll leave it for later, but you can get a hint by realizing that the units of a_0 are units of velocity.

Now, let's return to Euler's equation; substituting in the pressure and density deviations, it becomes

$$(\rho_0 + \rho_1) \left(\frac{\partial \mathbf{u}_1}{\partial t} + \mathbf{u}_1 \cdot \nabla \mathbf{u}_1 \right) = -\nabla(p_0 + p_1).$$

This can be simplified quite a bit – any time two small quantities are multiplied together they'll be dropped, and the pressure p_0 on the right hand side is constant, so its derivative will be zero. We're left with the much better

$$\rho_0 \frac{\partial \mathbf{u}_1}{\partial t} = -\nabla p_1. \quad (5.64)$$

Next up is the continuity equation, which becomes

$$\frac{\partial}{\partial t} (\rho_0 + \rho_1) + \nabla \cdot [(\rho_0 + \rho_1) \mathbf{u}_1] = 0$$

and simplifies to

$$\frac{\partial \rho_1}{\partial t} + \rho_0 \nabla \cdot \mathbf{u}_1 = 0. \quad (5.65)$$

Under the assumption of a small disturbance in an initially static fluid, we have been able to linearize the equations that describe compressible fluids. The response of the fluid is given by equations (5.63), (5.64), and (5.65). We just have a little bit more work to do – these are coupled equations and we need to uncouple them.

Here's how we do that. First, take the divergence of equation (5.64):

$$\rho_0 \frac{\partial}{\partial t} (\nabla \cdot \mathbf{u}_1) = -\nabla^2 p_1.$$

From equation (5.65), the divergence is

$$\nabla \cdot \mathbf{u}_1 = -\frac{1}{\rho_0} \frac{\partial \rho_1}{\partial t},$$

and plugging this into the equation above and simplifying gives

$$\frac{\partial^2 \rho_1}{\partial t^2} = \nabla^2 p_1.$$

Finally, using equation (5.63) to replace the density, we have a single equation for the pressure:

$$\frac{\partial^2 p_1}{\partial t^2} = a_0^2 \nabla^2 p_1. \quad (5.66)$$

This is a famous differential equation called the (three dimensional) *wave equation*. As its name suggests, it describes a wave; in this case, a pressure wave. The constant a_0 we defined above is the *speed* of the wave; in air at standard pressure and temperature it takes on the value $a_0 \approx 334$ m/s. We're talking, of course, about *sound waves*.

Example 5.5 – Sound waves in one dimension. If $p_1 = p_1(x, t)$ only, the wave equation becomes

$$\frac{\partial^2 p_1}{\partial t^2} = a_0^2 \frac{\partial^2 p_1}{\partial x^2}.$$

As you can show in Problem 5.8, the general solution to this differential equation is

$$p_1(x, t) = f(x - a_0 t) + g(x + a_0 t), \quad (5.67)$$

where f and g are arbitrary functions. This solution describes a wave propagating to the right (f) and left (g) with speed a_0 and no change of shape. The fact that there's no change of shape means that sound waves are *dispersionless*, unlike water waves.

Example 5.6 – Spherical sound waves. Let's now suppose that the pressure only depends on the *distance* r from the disturbance. In that case, the wave is spherically symmetric, and the wave equation in spherical coordinates with no dependence on the angular variables is

$$\frac{\partial^2 p_1}{\partial t^2} = a_0^2 \frac{1}{r^2} \frac{\partial}{\partial r} \left(r^2 \frac{\partial p_1}{\partial r} \right). \quad (5.68)$$

We can solve this by defining a new variable, β , such that

$$p_1(r, t) = \frac{1}{r} \beta(r, t).$$

Working out the derivatives, the wave equation surprisingly reduces to

$$\frac{\partial^2 \beta}{\partial t^2} = a_0^2 \frac{\partial^2 \beta}{\partial r^2},$$

exactly the one-dimensional form from the example above. That means the solution is the same for β , as well, and we have the pressure given by

$$p_1(r, t) = \frac{1}{r} [F(r - a_0 t) + G(r + a_0 t)].$$

In this case, F represents an *outgoing* wave and G an *incoming*. For a spherically symmetric disturbance in the fluid, however, we can drop the wave coming in from infinity since it doesn't make sense physically, and we have

$$p_1(r, t) = \frac{1}{r} F(r - a_0 t). \quad (5.69)$$

This describes a spherical wave expanding outward at the sound speed a_0 , but there's one big difference from the one dimensional example – the amplitude of the pressure wave *decreases* with distance as $1/r$. This of course is well-known – sounds are harder to hear the further away you are.

Although the examples above dealt with the pressure wave, notice that the density acts the same way because of equation (5.63) – they’re proportional to each other, so as the pressure increases, so too does the density. We could just as easily describe sound waves as density waves within the fluid.

Problems

Problem 5.1 – Effects of finite depth, part I. Consider small-amplitude sinusoidal water waves, as in Section 5.1.4, but assume the water has a finite depth at $y = -h$. Ignore any surface tension effects.

(a) Show that the dispersion relation is

$$\omega^2 = gk \tanh kh. \quad (5.70)$$

(b) Calculate the phase speed c and group speed c_g . What is the fastest phase speed these waves can have?

(c) For shallow water, $h \ll \lambda$, show that these waves are non-dispersive. What is their speed?

Problem 5.2 – Effects of finite depth, part II.

(a) For the case of finite depth, as in Problem 5.1, find the velocity potential $\varphi(x, y, t)$.

(b) Assuming that it doesn’t deviate too much from a mean position (\bar{x}, \bar{y}) , find the path a fluid particle takes under the surface. Plot the paths for a few different values of mean depth \bar{y} .

Problem 5.3 – Gravity-capillary dispersion relation. Using the pressure condition with surface tension, equation (5.23), show that the dispersion relation for sinusoidal gravity-capillary waves is given by

$$\omega^2 = gk + \frac{Tk^3}{\rho}.$$

Calculate and plot the phase speed c and group speed c_g of these waves.

Problem 5.4 – Capillary wave packets. Show that for pure capillary waves the group speed c_g is *twice* the phase speed c . What does this suggest for the motion of an individual peak in a wave packet of capillary waves? This, by the way, is why the capillary waves generated by the pole in the stream (Figure 5.9) travel upstream.

Problem 5.5 – Adding two cosines. Consider a surface wave consisting of two cosine waves as in Example 5.2:

$$\eta(x, t) = A \cos(k_1 x - \omega_1 t) + A \cos(k_2 x - \omega_2 t).$$

Show that, if $k_1 = k - \Delta k$, $k_2 = k + \Delta k$, and $\omega_1 = \omega - \Delta\omega$, $\omega_2 = \omega + \Delta\omega$, you can write the shape of the wave as

$$\eta(x, t) = 2A \cos(\Delta k x - \Delta\omega t) \cos(kx - \omega t),$$

so that it consists of a single sinusoidal wave modulated by an envelope.

Problem 5.6 – Calculating the energy of a wave.

(a) Show that the kinetic energy of a wave, integrated over a single wavelength, is given by equation (5.39).

(b) Show that the potential energy of a wave, integrated over a single wavelength, is given by equation (5.42).

Problem 5.7 – Long time response of capillary waves. Following Example 5.4, suppose a small disturbance in a pond creates *capillary* waves. After a long time and far from the disturbance, show that the surface is given by the real part of

$$\eta(x, t) = A(x, t)e^{4ipx^3/27Tt^2}.$$

Plot the surface at two different times.

Problem 5.8 – Solving the wave equation. The one dimensional wave equation for the pressure perturbation $p_1(x, t)$ is

$$\frac{\partial^2 p_1}{\partial t^2} = a_0^2 \frac{\partial^2 p_1}{\partial x^2}.$$

Define two new variables,

$$\xi = x - a_0 t \quad \text{and} \quad \eta = x + a_0 t,$$

and show that the wave equation becomes

$$\frac{\partial^2 p_1}{\partial \xi \partial \eta} = 0.$$

Solve this by direct integration and show that the general solution to the wave equation is

$$p_1(x, t) = f(x - a_0 t) + g(x + a_0 t).$$

Chapter 6

Special Topics

6.1 Deriving the Navier-Stokes Equation

As we'll see, the Navier-Stokes equation follows directly from Newton's second law; however, because it depends on the *deformation* of fluid elements, we need a whole new framework to describe the forces at work.

6.1.1 The Stress Tensor and Vector

To start, let's consider a fluid element – I'll draw it as a cube for simplicity but it can be any shape (Figure 6.1). If we apply a force that's normal to a surface, we'd expect the fluid element to deform by stretching and thinning. On the other hand, if the *same* force, in the same direction, is applied to a surface that's parallel to it, the deformation will be very different – the force in this case is called a “shear force.” Evidently, the deformation of a fluid element depends not only on the force direction but also on the *surface* direction.

This, unfortunately, adds a complication to how we must describe the forces that deform a fluid element. There are three orthogonal directions that the force itself could have, and three orthogonal directions that surface it's applied to could have – that means we need nine different numbers to fully specify things. Figure 6.2 shows this graphically; the nine elements together are called the *stress tensor* and are labelled T_{ij} , where i specifies the direction of the force, and j the direction of the surface. The elements of the stress tensor are forces per unit area; for example, the element T_{xy} gives the stress, or force per unit area, acting in the \hat{x} direction on a surface with normal in the \hat{y} direction.

It's common to write the tensors using either “index notation,” like T_{ij} ,

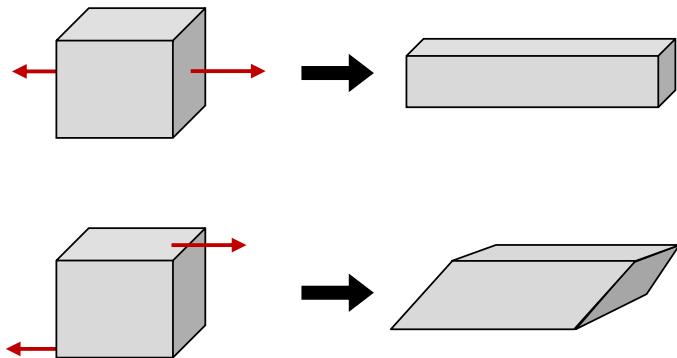


Figure 6.1: The deformation of a fluid element depends on both the force direction and the direction of the surface.

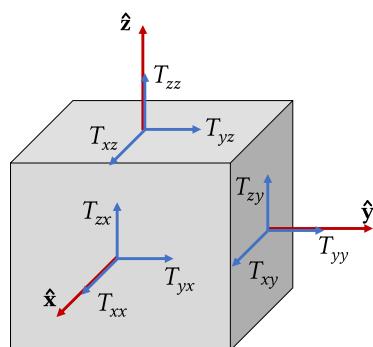


Figure 6.2: The definition of the stress tensor.

or as a matrix:

$$\mathbf{T} = \begin{bmatrix} T_{xx} & T_{xy} & T_{xz} \\ T_{yx} & T_{yy} & T_{yz} \\ T_{zx} & T_{zy} & T_{zz} \end{bmatrix}.$$

Note that the notation for a tensor, \mathbf{T} , is a little different from vectors. And it turns out that in almost all cases the stress tensor is *symmetric*, so that

$$T_{ij} = T_{ji},$$

which means that we really only have six elements to worry about rather than the full nine.

Suppose we have some surface S with a normal vector $\hat{\mathbf{n}}$; the *stress vector* is defined as

$$\mathbf{t} = \mathbf{T} \cdot \hat{\mathbf{n}}. \quad (6.1)$$

Careful here – that's a dot product between a tensor and a vector, not two vectors, and the result is a vector rather than a scalar. We can calculate the dot product either by direct summation,

$$t_i = \sum_{j=1}^3 T_{ij} n_j.$$

or by matrix multiplication,

$$\mathbf{t} = \begin{bmatrix} T_{xx} & T_{xy} & T_{xz} \\ T_{yx} & T_{yy} & T_{yz} \\ T_{zx} & T_{zy} & T_{zz} \end{bmatrix} \begin{bmatrix} n_x \\ n_y \\ n_z \end{bmatrix}.$$

As a check, let's suppose we have a surface where $\hat{\mathbf{n}} = \hat{\mathbf{x}}$. In that case, the stress vector will be

$$\mathbf{t} = \begin{bmatrix} T_{xx} & T_{xy} & T_{xz} \\ T_{yx} & T_{yy} & T_{yz} \\ T_{zx} & T_{zy} & T_{zz} \end{bmatrix} \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} T_{xx} \\ T_{yx} \\ T_{zx} \end{bmatrix},$$

which makes sense – the dot product picks out the forces that act on the surface with normal $\hat{\mathbf{x}}$.

6.1.2 Cauchy's Equation of Motion

Now we're ready to use Newton's second law to derive a general equation of motion for fluids and other deformable materials. Consider some "blob" of our fluid, with volume V and enclosed by surface S ; this blob can

deform in shape as the fluid moves, but we'll assume our fluid is incompressible, so the total volume can't change. The force across one small element of area, da , due to the fluid above it, is given by the stress vector \mathbf{t} multiplied by the area da . The total force acting on the entire blob due to the surrounding fluid is therefore

$$\mathbf{F} = \int_S \mathbf{t} da = \int_S (\mathbf{T} \cdot \hat{\mathbf{n}}) da, \quad (6.2)$$

where I used equation 6.1 to write the force in terms of the stress tensor instead of the stress vector. Now, we can apply the divergence theorem to turn this into an integral over the volume $d\tau$ – even though this is a tensor, it works pretty much the same way:

$$\mathbf{F} = \int_S (\mathbf{T} \cdot \hat{\mathbf{n}}) da = \int_V (\nabla \cdot \mathbf{T}) d\tau.$$

So far, we've ignored any forces external to the fluid, but we can add them in now. I'll stick with gravity; for a small volume of the fluid, the force of gravity will be $\rho d\tau g$, and we can combine this with the stress force above to get

$$\mathbf{F} = \int_V (\nabla \cdot \mathbf{T} + \rho g) d\tau.$$

That's the total force acting on our fluid, and we already know how to write out the acceleration – that was equation 1.10,

$$\frac{D\mathbf{u}}{Dt} = \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u}.$$

Newton's second law then gives

$$\int_V \rho \frac{D\mathbf{u}}{Dt} d\tau = \int_V (\nabla \cdot \mathbf{T} + \rho g) d\tau.$$

But this equation holds for *any* volume of any size within our fluid, so the integrands must be equal:

$$\rho \frac{D\mathbf{u}}{Dt} = \nabla \cdot \mathbf{T} + \rho g. \quad (6.3)$$

This is called Cauchy's equation of motion.

6.1.3 Newtonian Fluids

So far we've been very general with the stress tensor; specifying different tensors gives us different fluids modelled with Cauchy's equation. As a first example of a specific stress tensor – the simplest possible one! – consider

$$T_{ij} = -p \delta_{ij}, \quad (6.4)$$

where $p(x, y, z, t)$ is a scalar function – it's the *pressure* in the fluid – and δ_{ij} is called the Kronecker delta. If you're not familiar with it, this is a handy function (technically, it's a tensor) that is *one* if the two indices are the same, and *zero* if they're different:

$$\delta_{ij} = \begin{cases} 0 & i \neq j, \\ 1 & i = j \end{cases}.$$

As a matrix, this stress tensor therefore looks like

$$\boldsymbol{T} = \begin{bmatrix} -p & 0 & 0 \\ 0 & -p & 0 \\ 0 & 0 & -p \end{bmatrix}. \quad (6.5)$$

This stress tensor defines *ideal fluids*, which are inviscid (no viscosity) and don't include forces of deformation in the stress tensor. Why the negative sign on the pressure? Because this is the force of the fluid *outside* the surface pushing on it – in the opposite direction of the normal vector $\hat{\mathbf{n}}$.

Although ideal fluids have made up a large part of our discussion of fluid dynamics, we're here to derive the Navier-Stokes equations, which deal instead with *Newtonian fluids* and which do include the deformation of fluid elements. To incorporate that, we'll add a term to the ideal fluid stress tensor:

$$T_{ij} = T_{ij}^I + T_{ij}^D, \quad (6.6)$$

where the I indicates the ideal part and the D indicates the part that deals with the deformation.

Consider, for a moment, a *solid* instead of a fluid. If the solid is elastic, then pushing on it will cause it to deform in shape, and the stress that results will be proportional to the change in shape. This makes sense – it's just like in Hooke's law, where the force of the spring is proportional to the displacement. A fluid, though, is very different – pushing on a fluid causes deformation, but the deformation continues even when the force is stopped. It turns out that for fluids, the stress is proportional to the *velocities* of the change in shape; that is, rather than

$$t \propto dx,$$

we have

$$t \propto du/dx,$$

for example. Note a couple of things here: first, there are three velocities, u, v, w , and three directions, x, y, z , so we have nine different velocity gradients we can form. And second, note that these are derivatives with respect to *position*, not time; these are not accelerations.

Now, the most general symmetric tensor that depends *linearly* on these velocity gradients can be written as

$$T_{ij}^D = \mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right), \quad (6.7)$$

where μ is the constant of proportionality and is called (of course) the *viscosity*. Splitting the velocity gradients into two terms like this ensures the tensor is symmetric. Although it's cumbersome to do so, and doesn't really gain us anything, I'll write out this tensor in matrix form (you can at least compare with the ideal case in equation 6.5):

$$\mathbf{T}^D = \begin{bmatrix} 2\mu\partial u/\partial x & \mu(\partial v/\partial x + \partial u/\partial y) & \mu(\partial w/\partial x + \partial u/\partial z) \\ \mu(\partial u/\partial y + \partial v/\partial x) & 2\mu\partial v/\partial y & \mu(\partial w/\partial y + \partial v/\partial z) \\ \mu(\partial u/\partial z + \partial w/\partial x) & \mu(\partial v/\partial z + \partial w/\partial y) & 2\mu\partial w/\partial z \end{bmatrix}.$$

Adding in the ideal component, the total stress tensor is

$$T_{ij} = -p\delta_{ij} + \mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right). \quad (6.8)$$

This defines a *Newtonian fluid*. But you should know that there's plenty of examples of *non-Newtonian fluids*, as well – paint and blood are two good examples. Non-Newtonian fluids can be nonlinear in the velocity gradients, and can sometimes depend not just on the current state of the fluid, but on the past state as well – some of these fluids can have a “memory.” These kinds of fluids are well outside the scope of this book, however.

Example 6.1 – Plane parallel shear flow. Recall our example from Section 2.2.3 – we had an impulsively moved boundary, which put the fluid into plane parallel shear flow given in general by $u(y)$. For simplicity, we'll suppose the boundary is at $y = 0$ and the flow exists above it. What is the stress on the boundary from the fluid?

For flow of this type, the stress tensor becomes

$$\mathbf{T} = \begin{bmatrix} -p & \mu \partial u / \partial y & 0 \\ \mu \partial y / \partial y & -p & 0 \\ 0 & 0 & -p \end{bmatrix}, \quad (6.9)$$

and we can calculate the stress vector using equation 6.1, with $\hat{\mathbf{n}} = \hat{\mathbf{y}}$ for the boundary surface:

$$\mathbf{t} = \begin{bmatrix} -p & \mu \partial u / \partial y & 0 \\ \mu \partial y / \partial y & -p & 0 \\ 0 & 0 & -p \end{bmatrix} \begin{bmatrix} 0 \\ 1 \\ 0 \end{bmatrix}.$$

Carrying out the matrix multiplication gives

$$\mathbf{t} = \mu \frac{\partial u}{\partial y} \hat{\mathbf{x}} - p \hat{\mathbf{y}}.$$

This result makes sense: there's a viscous force along the boundary caused by friction with the fluid, and a force pushing downward on the surface from the fluid above.

We're finally ready to derive the Navier-Stokes equation, which is just Cauchy's equation of motion with the stress tensor for Newtonian fluids. I'm going to write out the calculation in index notation, which is a little easier to deal with, starting with Cauchy's equation,

$$\rho \frac{D u_i}{D t} = \sum_{j=1}^3 \frac{\partial T_{ij}}{\partial x_j} + \rho g_i$$

(compare with the vector version in equation 6.3). The only tricky thing here is the divergence of the stress tensor,

$$\sum_{j=1}^3 \frac{\partial T_{ij}}{\partial x_j} = \sum_{j=1}^3 \frac{\partial}{\partial x_j} \left[-p \delta_{ij} + \mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \right],$$

which is three derivatives. We'll tackle each separately.

First is the pressure term,

$$\sum_{j=1}^3 \frac{\partial}{\partial x_j} (-p \delta_{ij}) = - \sum_{j=1}^3 \frac{\partial p}{\partial x_j} \delta_{ij} = - \frac{\partial p}{\partial x_i}.$$

Notice that the Kronecker delta *collapsed* the sum – every term was zero except for the one where $j = i$.

The next derivative is

$$\sum_{j=1}^3 \frac{\partial}{\partial x_j} \left(\mu \frac{\partial u_j}{\partial x_i} \right) = \mu \frac{\partial}{\partial x_i} \sum_{j=1}^3 \frac{\partial u_j}{\partial x_j},$$

where I took the derivative with respect to x_i outside the sum (which is over j). Now, the term in the sum is familiar, even if it's hidden a bit – it's the divergence of the fluid velocity:

$$\sum_{j=1}^3 \frac{\partial u_j}{\partial x_j} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \nabla \cdot \mathbf{u}.$$

But we're assuming an incompressible flow, so the divergence is *zero*, and this derivative goes away.

Finally, we have the third derivative,

$$\sum_{j=1}^3 \frac{\partial}{\partial x_j} \left(\mu \frac{\partial u_i}{\partial x_j} \right) = \mu \sum_{j=1}^3 \frac{\partial^2 u_i}{\partial x_j^2},$$

and putting all three together gives us

$$\rho \frac{D u_i}{D t} = - \frac{\partial p}{\partial x_i} + \mu \sum_{j=1}^3 \frac{\partial^2 u_i}{\partial x_j^2} + \rho g_i.$$

Dividing both sides by the density ρ (and remembering that the kinematic viscosity is $\nu = \mu/\rho$) and using vector notation gives us, at long last, the Navier-Stokes equation,

$$\frac{D \mathbf{u}}{D t} = - \frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{g}.$$

(6.10)

6.2 Boundary Layers

The presence of a boundary layer in a fluid is one of the big differences between viscous flow and inviscid. To explore boundary layers further, let's set up a fairly simple situation. Consider fluid in the region $y \geq 0$, above a rigid boundary along $y = 0$. Suppose the flow is given by

$$\mathbf{u} = [\alpha x, -\alpha y, 0], \quad (6.11)$$

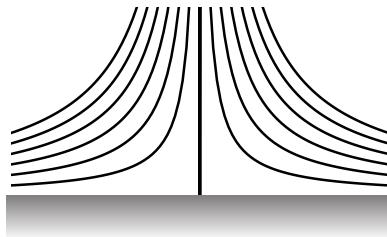


Figure 6.3: Streamlines for flow about a stagnation point. The fluid lies above the x -axis.

where α is some constant. We've seen this type of flow before, in Chapter 1 – it's the flow about a stagnation point. The streamlines are shown in Figure 6.3.

Now, this flow satisfies the Navier-Stokes equation, with a pressure given by

$$p(x, y) = -\frac{1}{2}\rho\alpha^2(x^2 + y^2) + \text{constant}. \quad (6.12)$$

However, it *doesn't* satisfy the no-slip boundary condition – the fluid velocity isn't zero at the boundary! This is actually fine if we're dealing with an inviscid fluid, where we expect the flow to be directed along the boundary, but doesn't work for a viscous fluid. It also has no indications of a boundary layer.

We need to fix things up, then, to account for the boundary layer and viscosity. We can do that by changing our flow slightly to

$$\mathbf{u} = [\alpha x f'(\eta), -\sqrt{\nu\alpha} f(\eta), 0], \quad (6.13)$$

where

$$\eta = \sqrt{\frac{\alpha}{\nu}} y$$

is a dimensionless length parameter. Here, the function $f(\eta)$ is an unknown function that we'll solve for shortly; notice that if $f(\eta) = \eta$, we get back our original flow about a stagnation point. However, in order to satisfy the no-slip condition and get zero velocity at $y = 0$, we need $f \rightarrow 0$ and $f' \rightarrow 0$ when $\eta \rightarrow 0$. Clearly $f(\eta) = \eta$ won't do the trick. On the other hand, we do expect the flow far away from the boundary to look like our original stagnation point flow; we'll return to this later.

To find the function $f(\eta)$ properly, then, we need to make sure that this flow still satisfies the Navier-Stokes equation. If we plug in the veloc-

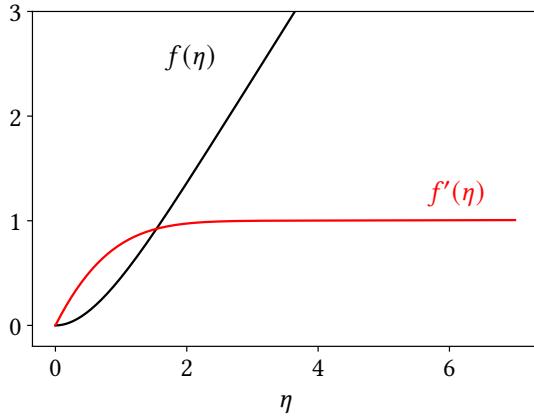


Figure 6.4: The function $f(\eta)$ (black) and $f'(\eta)$ (red). Note that the boundary layer is clear – it extends out to around $\eta \sim 2$. After that, $f(\eta) \approx \eta$.

ity in Equation (6.13) into the Navier-Stokes equation, the x and z equations become

$$\alpha^2 x f'^2 - \alpha^2 x f f'' = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \alpha^2 x f''' \quad (6.14)$$

and

$$\sqrt{\alpha^3 \nu} f f' = \frac{1}{\rho} \frac{\partial p}{\partial y} - \sqrt{\alpha^3 \nu} f'', \quad (6.15)$$

respectively. Let's rearrange the second equation slightly to get

$$-\frac{1}{\sqrt{\alpha^3 \nu}} \frac{1}{\rho} \frac{\partial p}{\partial y} = \frac{1}{2} \frac{d}{d\eta} f^2 + \frac{d}{d\eta} f'.$$

We can then replace the ∂y derivative with $\partial \eta$, since

$$\frac{d\eta}{dy} = \sqrt{\frac{\alpha}{\nu}},$$

to get

$$-\frac{1}{\alpha \nu \rho} \frac{\partial p}{\partial \eta} = \frac{1}{2} \frac{d}{d\eta} f^2 + \frac{d}{d\eta} f'.$$

Finally, integrate with respect to η :

$$-\frac{1}{\alpha \nu \rho} p = \frac{1}{2} f^2 + f' + g(x),$$

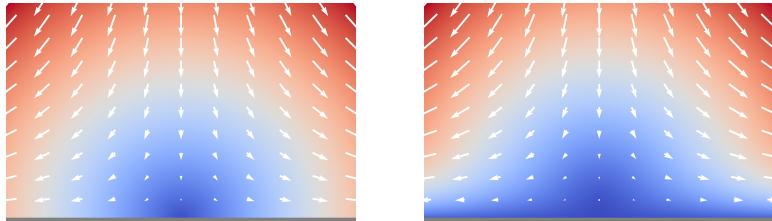


Figure 6.5: Vector plots of the inviscid flow (left) and viscous flow (right). The boundary layer can be clearly seen in the flow – the blue colour indicates small velocity.

where $g(x)$ is the integration “constant.” This is a useful result: if we now take the derivative with respect to x , we can see that

$$\frac{\partial p}{\partial x} \propto \frac{\partial g}{\partial x}.$$

In other words, $\partial p / \partial x$ is at best a function of x .

We can now return to the other Navier-Stokes equation, Equation (6.14), and separate variables so that all x terms are on one side and all η terms are on the other:

$$-\frac{1}{\alpha^2 \rho x} \frac{\partial p}{\partial x} = f'^2 - ff'' - f'''.$$

Since x and η (which is really just y) are independent, each side of this equation must be a constant. What is the value of that constant? To find it, we’ll compare with the inviscid solution for the pressure, Equation (6.12). To have our pressures agree, we need the constant to be one.

Finally, then, we have our last equation we need to solve,

$$f''' + ff'' - f'^2 + 1 = 0. \quad (6.16)$$

As we’ve discussed above, the boundary conditions are

$$f(0) = 0; \quad f'(0) = 0; \quad \text{and} \quad f'(\infty) = 1,$$

where the last condition ensures that we return to our original flow (with $f(\eta) = \eta$) far away from the boundary.

This equation must be solved numerically, but it can be tricky to do so while making sure the boundary conditions match; the results are shown in Figures 6.4 and 6.5. As you can see, there is now a small boundary layer where the velocity drops to zero, but away from the boundary the

flow is indistinguishable from the inviscid case. Despite the complexity of this example, it's a good one for exploring exactly how the boundary layer develops for viscous flows.

6.3 Nonlinear Waves

In our discussion of waves, we were careful to focus on those of small-amplitude – which linearized the equations that describe waves and allowed us to investigate various different properties. In general, the full theory of waves is quite difficult, but if we limit ourselves to shallow water, we can handle some interesting cases where the amplitude is large.

6.3.1 The Shallow Water Equations

We'll follow the basic set up we had back in Chapter 5, but make a few small changes. First, rather than using η for the height of the wave, I'll use $y = h(x, t)$ – mostly to remind us that these are not small amplitude waves. And we'll put a flat bottom at $y = 0$; see Figure 6.6. Furthermore, we won't assume that the amplitude of the wave is small compared to the wavelength. Instead, we'll make the *shallow water approximation* – that the depth of the water is much less than typical horizontal scales of the waves themselves.

As with the earlier work on waves, Euler's equation in two dimensions is the starting point, with $\mathbf{g} = [0, -g, 0]$:

$$\frac{Du}{Dt} = -\frac{1}{\rho} \frac{\partial p}{\partial x}, \quad (6.17)$$

$$\frac{Dv}{Dt} = -\frac{1}{\rho} \frac{\partial p}{\partial y} - g. \quad (6.18)$$

But the shallow water approximation suggests we can drop the Dv/Dt term from the second equation – basically, the acceleration in the vertical direction will be much less than the acceleration due to gravity, and can be neglected. Equation (6.18) then becomes

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

which integrates to

$$p(x, y, t) = -\rho gy + f_1(x, t),$$

where $f_1(x, t)$ is the integration “constant” (which could depend on x or t since we had a partial derivative). We can find the constant through

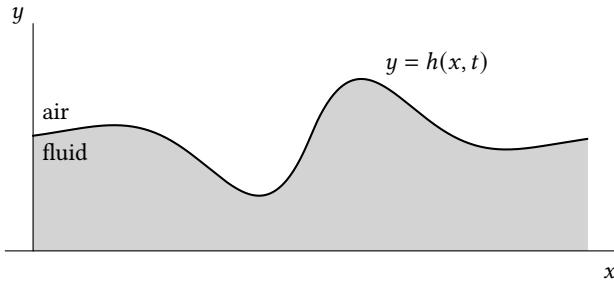


Figure 6.6: The surface of the fluid is given by $y = h(x, t)$, with the bottom at $y = 0$. These waves can be large in amplitude but the fluid itself is shallow.

boundary conditions as usual; in this case we have atmospheric pressure p_0 at the surface, $y = h(x, t)$. This leads to $f_1 = p_0 + \rho gh$, giving us the pressure

$$p(x, y, t) = p_0 - \rho g[y - h(x, t)]. \quad (6.19)$$

If we substitute this pressure into equation (6.17), we get

$$\frac{Du}{Dt} = -g \frac{\partial h}{\partial x}.$$

Now, h is a function of x and t only, which suggests that the *change* in u depends only on those variables. As long as u doesn't start out depending on y , it will never pick up a y dependency as time goes on. That means we can write $u = u(x, t)$, and, writing out the full material derivative and dropping the $\partial u / \partial y$ term, the above equation becomes

$$\boxed{\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} = -g \frac{\partial h}{\partial x}.} \quad (6.20)$$

This is the first of two important equations we'll use to describe nonlinear shallow waves. The second comes from the incompressibility condition, which in two dimensions says

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

If we integrate this for v , we get

$$v(x, y, t) = -\frac{\partial u}{\partial x}y + f_2(x, t),$$

where f_2 is again an integration constant. To find this one, we use the boundary condition at the bottom: we must have $v = 0$ at $y = 0$, which means $f_2 = 0$, and

$$v = -\frac{\partial u}{\partial x} y. \quad (6.21)$$

In addition to this, we can bring in the kinematic condition that we derived back in Section 5.1.2. Before linearizing, it was (replacing η with h)

$$v = \frac{\partial h}{\partial t} + u \frac{\partial h}{\partial x} \quad \text{at} \quad y = h(x, t).$$

Combining this with equation (6.21) above and rearranging a bit gives

$$\frac{\partial h}{\partial t} + u \frac{\partial h}{\partial x} + h \frac{\partial u}{\partial x} = 0.$$

(6.22)

Equations (6.20) and (6.22) are called the *shallow water equations*, and are a set of two coupled partial differential equations for the functions $u(x, t)$ (which is the horizontal speed of the fluid) and $h(x, t)$ (which is the height of the surface).

6.3.2 Method of Characteristics

To solve these shallow water equations, we'll apply a technique called the method of characteristics, which will reduce the two partial differential equations to two ordinary differential equations. We'll do that by finding a set of *characteristic curves*.

To begin, define a new variable $c(x, t)$ as

$$c(x, t) = \sqrt{gh(x, t)}. \quad (6.23)$$

This new variable is essentially a scaled height of the wave but will help us in making the two shallow water equations more symmetric. In terms of c , they become

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + 2x \frac{\partial c}{\partial x} = 0 \quad (6.24)$$

and

$$2 \frac{\partial c}{\partial t} + 2u \frac{\partial c}{\partial x} + c \frac{\partial u}{\partial x} = 0. \quad (6.25)$$

If we add these two equations together, we can write the result (with a bit of rearranging) as

$$\left[\frac{\partial}{\partial t} + (u + c) \frac{\partial}{\partial x} \right] (u + 2c) = 0. \quad (6.26)$$

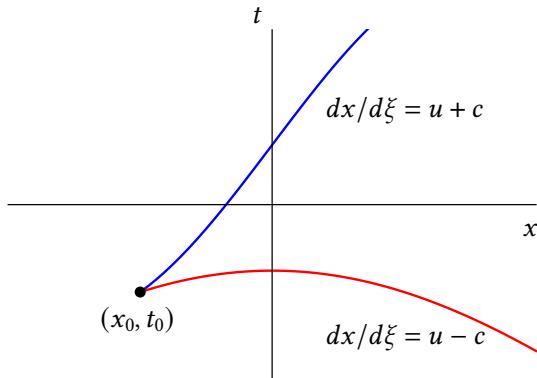


Figure 6.7: The positive characteristic curve (blue) is defined by $dx/d\xi = u + c$, and along the curve the quantity $u + 2c$ is constant. The negative characteristic curve (red) is defined by $dx/d\xi = u - c$, and along the curve the quantity $u - 2c$ is constant.

Similarly, if we subtract them, we get

$$\left[\frac{\partial}{\partial t} + (u - c) \frac{\partial}{\partial x} \right] (u - 2c) = 0. \quad (6.27)$$

These equations look pretty symmetric and are in good shape to use the method of characteristics on. Here's how that's done – but be careful, I think this is a tricky argument so try to follow along.

Consider just the positive equation (6.26) above. Define a parametric curve – called the *positive characteristic curve* – in the x - t plane using the parameter ξ , so that

$$x = x(\xi) \quad \text{and} \quad t = t(\xi).$$

The curve will be defined by two properties: that

$$\frac{dt}{d\xi} = 1,$$

(so that the parameter ξ is proportional to the time) and that

$$\frac{dx}{d\xi} = u + c.$$

We'll start the curve at some point (x_0, t_0) (see the blue line in Figure 6.7).

Why this definition? So that we can write the positive equation (6.26) as

$$\left[\frac{dt}{d\xi} \frac{\partial}{\partial t} + \frac{dx}{d\xi} \frac{\partial}{\partial x} \right] (u + 2c) = 0.$$

Now the term in the square brackets looks like a chain rule, meaning we can write this as simply

$$\boxed{\frac{d}{d\xi} (u + 2c) = 0.} \quad (6.28)$$

What does this mean? As the positive characteristic curve is traced out, the quantity $u + 2c$ stays *constant*; see the blue curve in Figure 6.7. Notice that we've reduced our original partial differential equation to an ordinary one; the penalty is that this equation is true only along the curve defined by $dx/d\xi = u + c$.

The whole argument above can be repeated with the negative equation (6.27), leading to a characteristic curve defined by $dx/d\xi = u - c$, and along this curve

$$\boxed{\frac{d}{d\xi} (u - 2c) = 0.} \quad (6.29)$$

Although the details at this point might be a little difficult to follow, the method of characteristics can be a powerful tool. A classic example should illustrate what's going on nicely.

6.3.3 The Dam Break Problem

Suppose a large dam holds back water of height h_0 as shown in Figure 6.8. The dam is located at $x = 0$, and the water is to the left, but at $t = 0$, the dam breaks – in our case, that means it disappears instantly – and the water can rush out to the right. We want to find two things: the horizontal speed of the water $u(x, t)$ and its height $h(x, t)$. This is a famous problem in fluid dynamics, not least because of the deadly and destructive nature of dam breaks, and was first solved in 1892 by Ritter.

Consider the state of the water *before* the dam breaks: for $x < 0$ and $t < 0$. Since the water is at rest, we know that $u = 0$ and $c = \sqrt{gh_0} = c_0$. We can start to build the characteristic curves with this information; let's do the positive curve first, defined by $dx/d\xi = u + c$. In this region, then, we have

$$\frac{dx}{d\xi} = c_0,$$

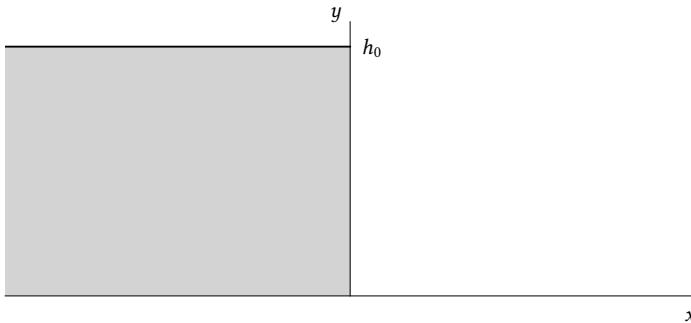


Figure 6.8: A dam at $x = 0$ holds back water of a height h_0 . At $t = 0$, the dam breaks and the water rushes forward.

or, since $dt/d\xi = 1$,

$$\frac{dx}{dt} = c_0.$$

Thus the positive characteristic curves in $x < 0, t < 0$ are straight lines of slope $c_0 = \sqrt{gh_0}$. I've plotted a few of these curves in Figure 6.9 (a) (the blue lines); each has the same slope but different intercepts. We can use similar reasoning for the negative characteristic curves to find that they have a slope of $-c_0$ (the red lines).

Now, we know that along the positive curves the quantity $u + 2c$ is constant. But in this region we know that $u + 2c = 2c_0$ since $u = 0$; similarly for the negative curves: $u - 2c = -2c_0$ along them. Here's the trick: this is true even for those curves that extend *beyond* the region $x < 0, t < 0$.

For example, take a look at point P_1 in Figure 6.9 (a) – it happens to be at an intersection of the two curves shown as dashed lines, one positive and one negative. Now, I don't know that those characteristic curves are still straight lines – when $t > 0$ the fluid has started to move, so our analysis above no longer holds. But we do know that along the positive line

$$u + 2c = 2c_0$$

and along the negative

$$u - 2c = -2c_0.$$

Right at point P_1 , both of these conditions are true, and we can rearrange and solve for u and c there. It turns out that

$$u = 0 \quad \text{and} \quad c = c_0$$

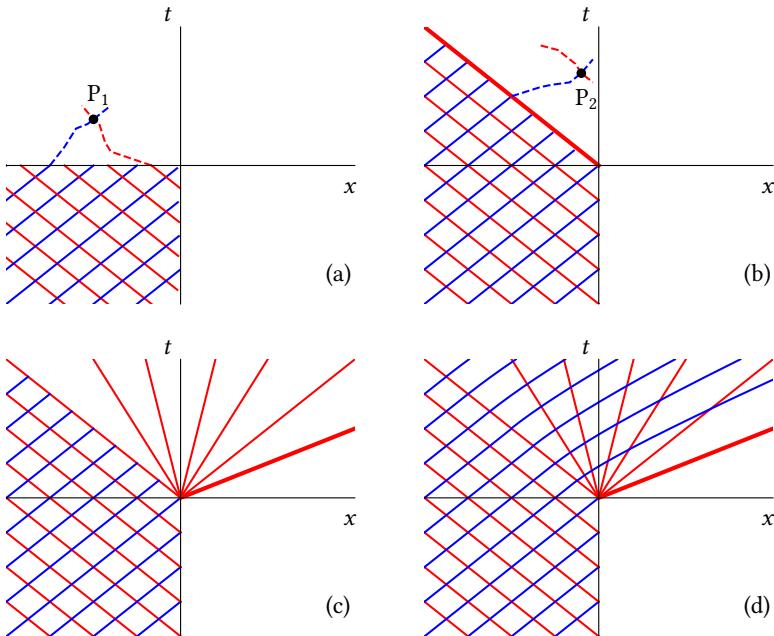


Figure 6.9: Constructing the characteristic curves for the dam break problem.

at point P_1 – but that means the lines *are* straight there, and will be throughout the region where the positive and negative curves intersect.

That's where the next twist comes in – the negative curves can't cover the entire second quadrant, since they must start from the third where the fluid is at rest. So the last negative characteristic curve is given by

$$x = -c_0 t,$$

and all curves below this line – the thick red line in Figure 6.9 (b) – are straight. That means for positions less than $-c_0 t$ the fluid is *still* at rest; this is to the left of the original dam location, and evidently the knowledge that the dam is no longer there has not yet been communicated beyond this point and the water there is undisturbed.

Beyond this line, though, things get a little more complicated. Consider the point P_2 in Figure 6.9 (b). It lies along a positive characteristic curve that originally started in the third quadrant, so even though this line is no longer straight, it still has

$$u + 2c = 2c_0$$

along it. There will be some negative curve passing through this point as well – but we don't know anything about this one other than $u - 2c$ is constant along it. Let's call the constant k , so that

$$u - 2c = k$$

along this negative curve. At point P_2 , then, we can use these two equations to solve for u and c ; they are

$$u = c_0 + \frac{k}{2} \quad \text{and} \quad c = \frac{c_0}{2} - \frac{k}{4}.$$

The important thing to note is that along the negative characteristic curve, both u and c are constant (since k is constant along that curve; the same is *not* true for the positive curve). Once again we find that the negative characteristic curves are straight lines, this time with slope

$$\frac{dx}{dt} = u - c = \frac{c_0}{2} + \frac{k}{4}.$$

That's great, but – what do these straight lines actually look like? Well, with a bit of thought you might realize that the negative characteristic curves can never cross (and the same is true of the positive, although obviously positive and negative can cross). If they did intersect at some point, the water there would have two different values of u or c . Normally the velocity and height are single-valued functions, so this is not allowed – except at one point: at $x = 0, t = 0$ we have a discontinuity in the height of the water. Thus the negative characteristic curves *can* cross at that point. So although we can't specify the slope of these lines, we do know that they start at the origin. This is shown in 6.9 (c).

Let's quickly recap what we know in the region above the line $x = -c_0 t$. Along the negative curves,

$$\frac{dx}{dt} = u - c = \text{constant},$$

which we can integrate to get

$$x = (u - c)t$$

(the integration constant is zero since the line passes through the origin). Along the positive curves, we have

$$u + 2c = c_0.$$

If we rearrange these two equations to solve for u and c we get

$$u(x, t) = \frac{2c_0}{3} + \frac{2x}{3t} \quad \text{and} \quad c(x, t) = \frac{2c_0}{3} - \frac{x}{3t}. \quad (6.30)$$

This is the velocity and height in the region $x > -c_0 t$. But note that $c = \sqrt{gh}$ can never go negative – so we must have, from the above equation

$$\frac{2c_0}{3} > \frac{x}{3t},$$

or $x < 2c_0 t$. This gives us the furthest extent of the water rushing from the dam – as time goes on, the water travels further. This is shown as the thick red line in Figure 6.9 (c) and (d).

We're all finished at this point – the fluid is either undisturbed or described by equation (6.30) – but before summarizing, let's finish off the positive characteristic curves in the region $x > -c_0 t$. The slope of these curves is given by

$$\frac{dx}{dt} = u + c = \frac{4c_0}{3} + \frac{x}{3t},$$

and we can solve this differential equation to get

$$x(t) = c_1 x^{1/3} + 2c_0 x, \quad (6.31)$$

where c_1 is an integration constant and varies from positive curve to positive curve. These lines are shown in Figure 6.9 (d) and complete the characteristic curve diagram for the dam break problem.

In summary, then, the height of the surface of the water is given by

$$h(x, t) = \begin{cases} h_0, & x < -\sqrt{gh_0}t \\ \frac{1}{g} \left[\frac{2}{3} \sqrt{gh_0} - \frac{x}{3t} \right]^2, & -\sqrt{gh_0}t < x < 2\sqrt{gh_0}t \\ 0, & x > 2\sqrt{gh_0}t, \end{cases} \quad (6.32)$$

and the velocity is

$$u(x, t) = \begin{cases} 0, & x < -\sqrt{gh_0}t \\ \frac{2}{3} \sqrt{gh_0} + \frac{2x}{3t}, & -\sqrt{gh_0}t < x < 2\sqrt{gh_0}t. \end{cases} \quad (6.33)$$

The surface is plotted in Figure 6.10 for two different times.

Notice that the speeds for typical modern dam heights can get very large; in Figure 6.10, I used a height of $h_0 = 10$ m, and at 2 s after the break, the leading edge of the water ($x = 2c_0 t \sim 40$ m) is travelling around 20 m/s. For larger dams, which can have heights of well over 100 m, the speed is in the hundreds of kilometers per hour.

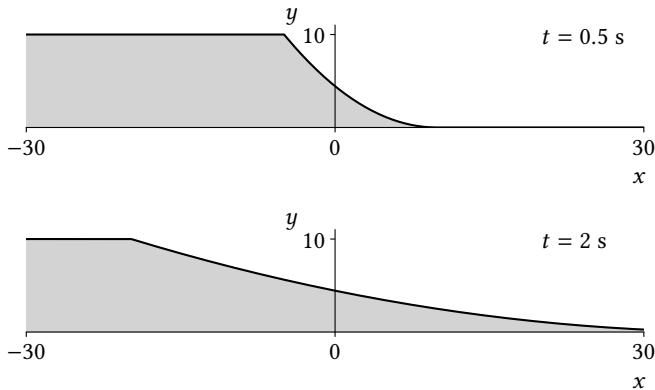


Figure 6.10: The height profile $h(x, t)$ for the water from a dam break at two different times. Distance units are in meters; at 2 s, the water has already reached 30 m from the original dam location.

6.4 Flow Past a Sphere

Throughout almost the entire book we've dealt only with flows that are two dimensional, but there's a classic problem involving flow around a *sphere* – decidedly not a two dimensional object – that's worth tackling. We'll start with the simpler case of ideal fluid flowing around a sphere, which is very similar to the cylinder problem we've already done in Section 3.2.2, and build up to the flow of very viscous fluid past a sphere.

6.4.1 Ideal Flow

Consider first, then, the ideal flow around a sphere of radius a . The fluid starts out, far away from the sphere, in uniform flow with speed U . We'll put the sphere at the origin and arrange our coordinate system such that the uniform flow is along the z axis, so far away $\mathbf{u} = [0, 0, U]$; see Figure 6.11.

Now, this flow is *irrotational* for the same reason ideal flow about a cylinder is – since the flow is uniform at infinity, it obviously has zero vorticity there, and Kelvin's circulation theorem (Section 3.1.4) says that it will remain irrotational as it flows past the sphere. That means we can describe the flow with a velocity potential φ as usual and, since the flow is incompressible, it must satisfy Laplace's equation,

$$\nabla^2 \varphi = 0.$$

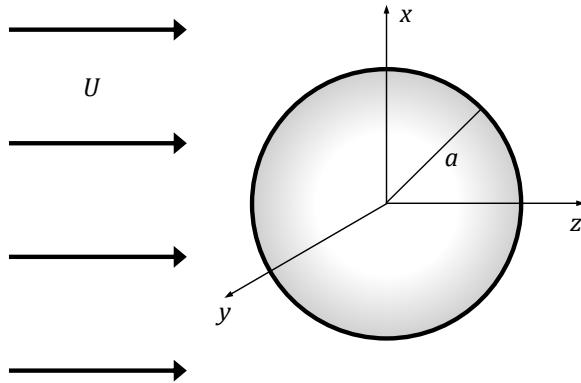


Figure 6.11: Fluid that is uniform at infinity flows past a sphere.

Before we start the heavy machinery of solving Laplace's equation via separation of variables, though, it's worthwhile to look at two things: whether there's any symmetry in the problem we can exploit, and what our boundary conditions are. We *do* in fact have symmetry here: since we could rotate the flow and sphere by any angle we wish around the z axis and not change the problem, there must not be any dependence on the spherical ϕ coordinate (that's the azimuthal angle, measured from the x axis around the z axis). This is called *axisymmetry*. In spherical coordinates, then, we can write

$$\mathbf{u} = \mathbf{u}(r, \theta), \quad (6.34)$$

and write Laplace's equation as

$$\nabla^2 \varphi = \frac{1}{r^2} \frac{\partial}{\partial r} \left(r^2 \frac{\partial \varphi}{\partial r} \right) + \frac{1}{r^2 \sin \theta} \frac{\partial}{\partial \theta} \left(\sin \theta \frac{\partial \varphi}{\partial \theta} \right). \quad (6.35)$$

The boundary conditions turn out to be similar to those for flow around a cylinder (Section 3.2.2 again). We need the fluid to slip along the boundary of the sphere, which means there can't be a radial component there, so the first boundary condition is

$$u_r(a, \theta) = 0 \quad \text{or} \quad \left. \frac{\partial \varphi}{\partial r} \right|_{r=a} = 0. \quad (6.36)$$

The second condition is that the flow far away from the sphere must be uniform along the z direction. In spherical coordinates, this uniform flow

is given by

$$u_r(r, \theta) = U \cos \theta \quad \text{and} \quad u_\theta(r, \theta) = -U \sin \theta. \quad (6.37)$$

From $\mathbf{u} = \nabla \varphi$, which in spherical coordinates is

$$u_r = \frac{\partial \varphi}{\partial r} \quad \text{and} \quad u_\theta = \frac{1}{r} \frac{\partial \varphi}{\partial \theta},$$

we get that the velocity potential for uniform flow is

$$\varphi_{\text{uni}} = Ur \cos \theta.$$

The boundary condition is therefore

$$\varphi(r, \theta) \rightarrow Ur \cos \theta \quad \text{as} \quad r \rightarrow \infty. \quad (6.38)$$

Laplace's equation in spherical coordinates with axisymmetry is a well-studied problem in numerous physics subjects; the angular part of the solution is the Legendre polynomials $P_\ell(\cos \theta)$. However, we can skip the full separation of variables if we examine that second boundary condition and suppose that our solution must take on the form

$$\varphi(r, \theta) = f(r) \cos \theta, \quad (6.39)$$

where $f(r)$ is some unknown function. This is just the $\ell = 1$ Legendre polynomial.

We can find $f(r)$ by plugging equation (6.39) into Laplace's equation as given in equation (6.35). After some work, it reduces to the ordinary differential equation

$$r^2 \frac{d^2 f}{dr^2} + 2r \frac{df}{dr} - 2f = 0. \quad (6.40)$$

This equation can be solved by guessing that the solution is a power-law, $f(r) = r^m$. Plugging this form in gives us the quadratic

$$m^2 + m - 2 = 0,$$

which has solutions $m = 1$ and $m = -2$. Thus the general solution is

$$f(r) = Ar + \frac{B}{r^2},$$

and the velocity potential is then

$$\varphi(r, \theta) = \left(Ar + \frac{B}{r^2} \right) \cos \theta.$$

Fitting the boundary conditions, equations (6.36) and (6.38), gives

$$A = U \quad \text{and} \quad B = \frac{Ua^3}{2},$$

so the final velocity potential for ideal flow around a sphere is

$$\varphi(r, \theta) = U \left(r + \frac{a^3}{2r^2} \right) \cos \theta. \quad (6.41)$$

Given the potential, we can find the velocity \mathbf{u} by taking the gradient; Problem 6.2 asks you to do it, and the result is

$$u_r(r, \theta) = U \left(1 - \frac{a^3}{r^3} \right) \cos \theta \quad (6.42)$$

and

$$u_\theta(r, \theta) = -U \left(1 + \frac{a^3}{2r^3} \right) \sin \theta. \quad (6.43)$$

Feel free to compare this solution with that for the cylinder in Section 3.2.2.

We could, at this point, calculate the pressure on the sphere and find the force that the fluid exerts on it. However, I'll skip the details here – I've shown the pressure throughout the fluid in Figure 6.12 if you're interested – and just mention that the end result is exactly the same as in the cylinder case: the net force is zero. This is just d'Alembert's paradox showing up again since we'll dealing with an ideal fluid.

6.4.2 Stokes Stream Function

How do we plot the streamlines for the fluid? For two dimensional, incompressible, irrotational flow we had the stream function ψ , which was constant on streamlines and made it simple to plot them. For axisymmetric flow, the same stream function no longer works. However, there is an analog for this case, called *Stokes stream function*¹ Ψ , defined such that

$$\mathbf{u} = \nabla \times \left(\frac{1}{r \sin \theta} \hat{\Psi} \right). \quad (6.44)$$

In spherical coordinates, this definition becomes

$$u_r = \frac{1}{r^2 \sin \theta} \frac{\partial \Psi}{\partial \theta} \quad \text{and} \quad u_\theta = -\frac{1}{r \sin \theta} \frac{\partial \Psi}{\partial r}. \quad (6.45)$$

¹The two dimensional stream function ψ is then sometimes called the Lagrange stream function. Then again, some places call both versions Stokes stream functions, or just stream functions, so be careful.

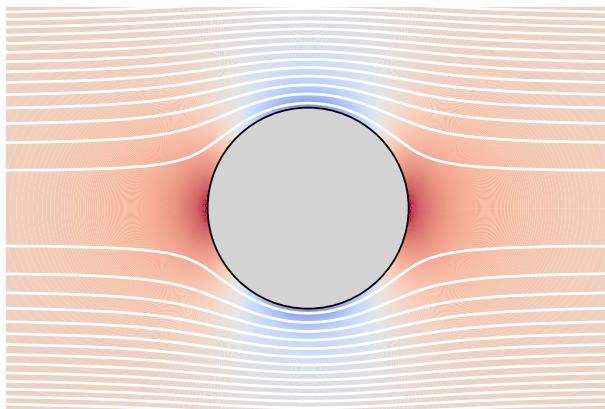


Figure 6.12: Streamlines for ideal flow about a sphere. The pressure is shown as well, with red regions indicating high pressure and blue low.

You can show for yourself (Problem 6.2) that with this definition and for axisymmetric flow,

$$(\mathbf{u} \cdot \nabla)\Psi = 0,$$

so that Ψ is indeed constant along streamlines as we wanted.

For the sphere, you can show that (Problem 6.2 again)

$$\Psi(r, \theta) = \frac{1}{2}U \left(r^2 - \frac{a^3}{r} \right) \sin^2 \theta. \quad (6.46)$$

The streamlines are plotted in Figure 6.12; again, you might want to compare with the streamlines for flow around a cylinder in Figure 3.6.

6.4.3 Very Viscous Flow

Suppose instead of ideal fluid flowing past the sphere, we have Newtonian viscous flow instead. If the flow is *steady* and we ignore the effect of gravity, the Navier-Stokes equation (2.1) is then

$$(\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\mathbf{u}. \quad (6.47)$$

There are two terms involving velocity in this equation, and their relative strength determines how a fluid will flow. We've already studied at length flows for which the *inertial term* $(\mathbf{u} \cdot \nabla)\mathbf{u}$ is dominant over the *viscous term* $\nu\nabla^2\mathbf{u}$ – that's just ideal fluid flow. Here we turn to the case

where the viscous term is much larger than the inertial, in which case we'll drop that term and equation (6.47) simply becomes

$$\nabla p = \mu \nabla^2 \mathbf{u} \quad (6.48)$$

(notice that this has the dynamic viscosity $\mu = \nu/\rho$ in it). This equation is of course supplemented as always by the incompressibility condition,

$$\nabla \cdot \mathbf{u}. \quad (6.49)$$

Together, these equations are sometimes called the *slow flow equations* – they are a good description of fluid flow that is *very viscous* or, alternatively, has a small Reynolds number,

$$R = \frac{UL}{\nu} \ll 1.$$

Before we proceed with slow flow past a sphere, I want to rewrite equation (6.48) to get rid of the pressure term. We can do that by using the vector identity

$$\nabla^2 \mathbf{u} = \nabla(\nabla \cdot \mathbf{u}) - \nabla \times (\nabla \times \mathbf{u}).$$

Plugging this into the slow flow equation (6.48), dropping the first term (since we have incompressible flow), and writing it in terms of the vorticity $\boldsymbol{\omega} = \nabla \times \mathbf{u}$, we get

$$\nabla p = -\mu \nabla \times \boldsymbol{\omega}. \quad (6.50)$$

Finally, if we take the curl of both sides, the left hand side becomes zero (since the curl of a gradient is identically zero; see Problem 1.4), and we get

$$\boxed{\nabla \times (\nabla \times \boldsymbol{\omega}) = \mathbf{0}.} \quad (6.51)$$

This is the equation we'll solve; it actually looks simpler than it is, since in terms of the velocity \mathbf{u} it has *three* curls.

6.4.4 Slow Flow Past a Sphere

Now we can attempt a solution for the case of very viscous flow past a sphere. This is a classic problem in fluid dynamics, first done by Stokes in 1851. It's not too difficult, but it is lengthy and a little complicated, so I'll go slow and carefully.

The set up will look just like the ideal case as shown in Figure 6.11: the fluid will be uniform at infinity with speed U , and our coordinate system will be such that the flow is along the z direction and the sphere of radius

a is at the origin. So, once again we have axisymmetry, and the velocity will not depend on ϕ at all:

$$\mathbf{u} = u_r(r, \theta)\hat{\mathbf{r}} + u_\theta(r, \theta)\hat{\mathbf{\theta}}.$$

Unfortunately, though, this problem is no longer irrotational, so Laplace's equation is no longer appropriate, and our similarity to the ideal case stops. However, we *do* still have incompressible, axisymmetric flow, so Stokes stream function $\Psi(r, \theta)$ can be used to describe the fluid. Equation (6.45) gives the velocity in terms of Ψ , and I'll rewrite it here so you don't have to flip back to see it:

$$u_r = \frac{1}{r^2 \sin \theta} \frac{\partial \Psi}{\partial \theta} \quad \text{and} \quad u_\theta = -\frac{1}{r \sin \theta} \frac{\partial \Psi}{\partial r}.$$

The Equation of Motion

Our first goal is to write the slow flow equation (6.51) in terms of the stream function Ψ . That's a lot of curls we have to work through, but the end result isn't too bad.

We can calculate the vorticity easily enough; since there's no u_ϕ or any ϕ dependence, the curl of \mathbf{u} will only have a component in the $\hat{\phi}$ direction. In spherical coordinates, this is

$$\boldsymbol{\omega} = \nabla \times \mathbf{u} = \frac{1}{r} \left[\frac{\partial(ru_\theta)}{\partial r} - \frac{\partial u_r}{\partial \theta} \right] \hat{\phi}. \quad (6.52)$$

Plugging in u_r and u_θ from above, this becomes, in terms of Ψ ,

$$\boldsymbol{\omega} = -\frac{1}{r \sin \theta} \left[\frac{\partial^2 \Psi}{\partial r^2} + \frac{\sin \theta}{r^2} \frac{\partial}{\partial \theta} \left(\frac{1}{\sin \theta} \frac{\partial \Psi}{\partial \theta} \right) \right] \hat{\phi}.$$

Now, this is a complicated expression, so we're going to use a little trick to make the subsequent work easier. It's not much of a trick: define an operator \mathbb{L}^2 as

$$\mathbb{L}^2 = \frac{\partial^2}{\partial r^2} + \frac{\sin \theta}{r^2} \frac{\partial}{\partial \theta} \left(\frac{1}{\sin \theta} \frac{\partial}{\partial \theta} \right), \quad (6.53)$$

so that the vorticity becomes

$$\boldsymbol{\omega} = -\frac{1}{\sin \theta} \mathbb{L}^2 \Psi \hat{\phi}.$$

We're just hiding some complexity inside of that new operator, but it turns out that this pattern will repeat.

Next we need to take the curl of the vorticity. With some tedious but not difficult work, you can show that it is

$$\nabla \times \omega = -\frac{1}{r^2 \sin \theta} \frac{\partial}{\partial \theta} (\mathbb{L}^2 \Psi) \hat{r} + \frac{1}{r \sin \theta} \frac{\partial}{\partial r} (\mathbb{L}^2 \Psi) \hat{\theta}. \quad (6.54)$$

Now we just need one more curl. Since $\nabla \times \omega$ only has r and θ directions and dependence, once again the curl will only have a $\hat{\phi}$ direction. It is

$$\nabla \times \nabla \times \omega = \frac{1}{r \sin \theta} \left[\frac{\partial^2}{\partial r^2} (\mathbb{L}^2 \Psi) + \frac{\sin \theta}{r^2} \frac{\partial}{\partial \theta} \left(\frac{1}{\sin \theta} \frac{\partial}{\partial \theta} (\mathbb{L}^2 \Psi) \right) \right] \hat{\phi}.$$

See the pattern? This is just

$$\nabla \times \nabla \times \omega = \frac{1}{r \sin \theta} \mathbb{L}^2 (\mathbb{L}^2 \Psi) \hat{\phi},$$

or, extending the notation in an obvious way,

$$\nabla \times \nabla \times \omega = \frac{1}{r \sin \theta} \mathbb{L}^4 \Psi \hat{\phi}.$$

We have, then, our equation of motion for very viscous fluid moving past a sphere; plugging this into equation (6.51), we have

$$\boxed{\mathbb{L}^4 \Psi = 0.} \quad (6.55)$$

This might *look* simple, but don't be fooled: this is a fourth order partial differential equation.

The Boundary Conditions

As with the ideal case above, looking at the boundary conditions can give us a hint on how to proceed with actually solving this fourth order equation. The boundary conditions are actually pretty similar to that ideal fluid problem.

First, we need to apply the no-slip condition to the surface of the sphere – the fluid must have zero velocity there, so

$$u_r(a, \theta) = 0 \quad \text{and} \quad u_\theta(a, \theta) = 0.$$

We can write this condition in terms of the stream function Ψ using equation (6.45) and get

$$\frac{\partial \Psi}{\partial \theta} \Big|_{r=a} = 0 \quad \text{and} \quad \frac{\partial \Psi}{\partial r} \Big|_{r=a} = 0. \quad (6.56)$$

Second, the flow needs to be uniform in the z direction far away from the sphere. That means we need

$$\mathbf{u}(r \rightarrow \infty) = U \hat{\mathbf{z}},$$

or, in spherical coordinates,

$$u_r \rightarrow U \cos \theta \quad \text{and} \quad u_\theta \rightarrow -U \sin \theta \quad \text{as} \quad r \rightarrow \infty.$$

In terms of the Stokes stream function, this boundary condition is

$$\Psi \rightarrow \frac{1}{2} Ur^2 \sin^2 \theta. \quad (6.57)$$

The Solution

This second boundary condition suggests that our solution will take the form

$$\Psi(r, \theta) = f(r) \sin^2 \theta. \quad (6.58)$$

This form of Ψ will simplify things quite a bit. Let's start by applying the operator \mathbb{L}^2 to this form of Ψ ; after a few lines of work, I get

$$\mathbb{L}^2 \Psi = \left(f'' - \frac{2}{r^2} f \right) \sin^2 \theta,$$

where the double prime indicates two derivatives with respect to r .

We can make the next step a little easier if we define a new function

$$F(r) = f'' - \frac{2}{r^2} f. \quad (6.59)$$

Then applying the \mathbb{L}^2 operator again obviously gives us the same result:

$$\mathbb{L}^2(\mathbb{L}^2 \Psi) = \left(F'' - \frac{2}{r^2} F \right) \sin^2 \theta,$$

just in terms of F rather than f . Equation (6.55),

$$\mathbb{L}^4 \Psi = 0,$$

becomes

$$F'' - \frac{2}{r^2} F = 0.$$

Plugging in equation (6.59) gives us

$$f'''' - \frac{4}{r^2} f'' + \frac{8}{r^3} f' - \frac{8}{r^4} f = 0. \quad (6.60)$$

As promised, this is a fourth order differential equation that we need to solve. However, this is easier than you might guess; first, multiply by r^4 to get

$$r^4 \frac{d^4 f}{dr^4} - 4r^2 \frac{d^2 f}{dr^2} + 8r \frac{df}{dr} - 8f = 0.$$

Notice that each term is paired with a similar order r and derivative; this suggests we try a power-law solution of the form

$$f(r) = r^m.$$

Plugging this into the differential equation above gives us a fourth-order polynomial,

$$(m-3)(m-2)(m-1)m - 4(m-1)m + 8m - 8 = 0,$$

which has the four roots

$$m = -1, 1, 2, \text{ and } 4.$$

Thus the general solution for $f(r)$ is

$$f(r) = \frac{A}{r} + Br + Cr^2 + Dr^4. \quad (6.61)$$

Now we just have to find the constants A, B, C , and D using the boundary conditions.

Let's start with the second, that the flow must be uniform at infinity – so $\Psi \sim r^2$ there. Of the four terms in the general solution, it doesn't really tell us anything about the first two, since the first (A) term goes to zero at infinity, and the second (B) doesn't get as big as r^2 so it can be neglected as $r \rightarrow \infty$. But it does tell us that we need to drop the fourth term by setting $D = 0$, since that term blows up faster than r^2 . Then, looking at the form of the stream function in equation (6.57), we must have for the third term

$$C = \frac{1}{2}U.$$

The first boundary condition, equation (6.56), is for the derivatives of Ψ to go to zero at $r = a$. I'll leave it to you to work out the derivatives themselves and show that

$$A = \frac{1}{4}Ua^3 \quad \text{and} \quad B = -\frac{3}{4}Ua.$$

Finally, after much work, we come to our solution at last: Stokes stream function for slow flow past a sphere is

$$\Psi(r, \theta) = U \left(\frac{a^3}{4r} - \frac{3a}{4}r + \frac{1}{2}r^2 \right) \sin^2 \theta. \quad (6.62)$$

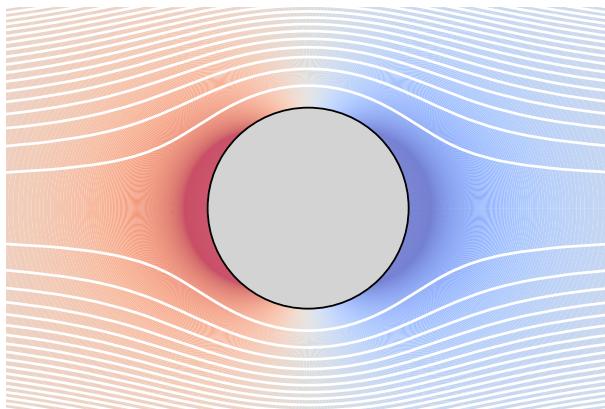


Figure 6.13: Streamlines for very viscous flow past a sphere. The colours indicate pressure, with red areas being high pressure.

From this we can immediately plot the streamlines of the flow (Figure 6.13), and we can find the velocities from equation (6.45),

$$u_r(r, \theta) = U \left(\frac{a^3}{2r^3} - \frac{3a}{2r} + 1 \right) \cos \theta, \quad (6.63)$$

and

$$u_\theta(r, \theta) = U \left(\frac{a^3}{4r^3} + \frac{3a}{4r} - 1 \right) \sin \theta. \quad (6.64)$$

6.4.5 Stokes Law

More interesting in this case is the *force* on the sphere from the fluid. In our previous examples of ideal fluid flow past objects, the force was always zero, called d'Alembert's paradox. But now that we have actual viscous fluid, there *will* be a force. Our goal is to calculate it, but be warned: we could get away with just integrating over the pressure in past examples, but with viscous fluids there are additional forces to take into account.

The Pressure

We'll start with the pressure, though, since we have a simple expression for it in equation (6.50),

$$\nabla p = -\mu \nabla \times \boldsymbol{\omega}.$$

In spherical coordinates (and ignoring the ϕ component since we have axisymmetry) this is two equations,

$$\frac{\partial p}{\partial r} = -\mu[\nabla \times \omega]_r, \quad (6.65)$$

$$\frac{1}{r} \frac{\partial p}{\partial \theta} = -\mu[\nabla \times \omega]_\theta, \quad (6.66)$$

where I'm sorry for the poor notation indicating the components of $\nabla \times \omega$. Those components are easy to find from equation (6.54), where

$$\mathbb{L}^2 \Psi = \frac{3Ua}{2r} \sin^2 \theta.$$

So

$$[\nabla \times \omega]_r = -\frac{3Ua}{r^3} \cos \theta$$

and

$$[\nabla \times \omega]_\theta = -\frac{3Ua}{2r^3} \sin \theta,$$

and the two pressure equations become

$$\frac{\partial p}{\partial r} = \frac{3U\mu a}{r^3} \cos \theta, \quad (6.67)$$

$$\frac{1}{r} \frac{\partial p}{\partial \theta} = \frac{3U\mu a}{2r^3} \sin \theta. \quad (6.68)$$

Integrating gives the pressure,

$$p(r, \theta) = p_\infty - \frac{3U\mu a}{2r^2} \cos \theta, \quad (6.69)$$

where p_∞ is the integration constant and is the pressure at infinity. Figure 6.13 shows the pressure field in the fluid surrounding the sphere, where red indicates high pressure and blue low. Of course, for the force calculation we'll need the pressure evaluated at the surface of the sphere $r = a$; this is

$$p(a, \theta) = p_\infty - \frac{3U\mu}{2a} \cos \theta. \quad (6.70)$$

As you can see in Figure 6.13, the pressure is a maximum at $\theta = \pi$ and a minimum when $\theta = 0$. This pressure difference upstream and downstream of the flow accounts for part of the force on the sphere (but not all of it). You should compare with Figure 3.7 for ideal flow past a cylinder, for which there is no force at all.

The Stress

In Section 6.1 I introduced the stress tensor \mathbf{T} , which is a collection of nine different quantities T_{ij} that quantify the force exerted by the fluid, including both the pressure and viscous forces. For Newtonian fluids, this stress tensor can be written as (see equation 6.8)

$$\mathbf{T} = \begin{bmatrix} -p + 2\mu\partial u/\partial x & \mu(\partial v/\partial x + \partial u/\partial y) & \mu(\partial w/\partial x + \partial u/\partial z) \\ \mu(\partial u/\partial y + \partial v/\partial x) & -p + 2\mu\partial v/\partial y & \mu(\partial w/\partial y + \partial v/\partial z) \\ \mu(\partial u/\partial z + \partial w/\partial x) & \mu(\partial v/\partial z + \partial w/\partial y) & -p + 2\mu\partial w/\partial z \end{bmatrix}.$$

Unfortunately, this is in Cartesian coordinates, and we're working with spherical – and the corresponding tensor in spherical coordinates is very messy. Luckily, as we'll see, we really only need two different components of the stress tensor: T_{rr} , the force in the radial direction against a surface with normal in the radial direction, and $T_{\theta r}$, the force in the $\hat{\theta}$ direction against a surface with normal in the radial direction. Deriving these two components is a little beyond the scope of this book,² but I can give them to you here:

$$T_{rr} = -p + 2\mu \frac{\partial u_r}{\partial r} \quad (6.71)$$

$$T_{\theta r} = \mu \left(\frac{1}{r} \frac{\partial u_r}{\partial \theta} + \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r} \right). \quad (6.72)$$

For our case, with the velocities given by equations (6.63) and (6.64) and the pressure by equation (6.69), the stress tensor components become

$$T_{rr}(r, \theta) = -p_\infty + \frac{9U\mu a}{2r^2} \cos \theta - \frac{6U\mu a^3}{2r^4} \cos \theta,$$

$$T_{\theta r}(r, \theta) = -\frac{3U\mu a^3}{2r^4} \sin \theta,$$

and, when evaluated at $r = a$, we have

$$T_{rr}(a, \theta) = -p_\infty + \frac{3U\mu}{2a} \cos \theta, \quad (6.73)$$

$$T_{\theta r}(a, \theta) = -\frac{3U\mu}{2a} \sin \theta. \quad (6.74)$$

²I looked them up for this calculation; see, for example, *Elementary Fluid Dynamics* by D. J. Acheson, appendix A.7.

The Force on the Sphere

We're now finally in a position to calculate the force on the surface of the sphere from the surrounding fluid. In terms of the stress tensor, this force is given by equation (6.2),

$$\mathbf{F} = \int_S (\mathbf{T} \cdot \hat{\mathbf{n}}) da.$$

The surface of the sphere has a normal $\hat{\mathbf{n}} = \hat{\mathbf{r}}$, and remember that the dot product like this between a tensor and a vector gives us another vector, not a scalar. This vector is the *stress tensor* (see Section 6.1.1), which here I'll call \mathbf{t}_r :

$$\mathbf{t}_r = \mathbf{T} \cdot \hat{\mathbf{r}}.$$

For clarity, I'll write out this calculation in matrix format so we can see exactly what is going on. The stress vector then has components

$$\mathbf{t}_r = \begin{bmatrix} T_{rr} & T_{r\theta} & T_{r\phi} \\ T_{\theta r} & T_{\theta\theta} & T_{\theta\phi} \\ T_{\phi r} & T_{\phi\theta} & T_{\phi\phi} \end{bmatrix} \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} T_{rr} \\ T_{\theta r} \\ T_{\phi r} \end{bmatrix}$$

In other words, the dot product picks out the components of the stress tensor that apply to a surface in the radial $\hat{\mathbf{r}}$ direction. The net force is then the surface integral over each component.

But we know from symmetry arguments that the net force on the sphere should be in the z direction, so we can shorten our work a bit by only considering the z component of this stress vector, in which case we want to calculate

$$F_z = \int_S (\mathbf{t}_r \cdot \hat{\mathbf{z}}) da.$$

In spherical coordinates, $\hat{\mathbf{z}} = \cos \theta \hat{\mathbf{r}} - \sin \theta \hat{\theta}$, so doing the dot product gives us

$$F_z = \int_S (T_{rr} \cos \theta - T_{\theta r} \sin \theta) da.$$

As mentioned above, we only need those two components of the stress tensor, coming from the direction of the surface ($\hat{\mathbf{r}}$) and the direction of the net force (in both the $\hat{\mathbf{r}}$ and $\hat{\theta}$ directions).

Plugging in the components T_{rr} and $T_{\theta r}$ (evaluated at $r = a$) and using $da = a^2 \sin \theta d\theta d\phi$ for the surface element of the sphere gives us the integral

$$F_z = \int_0^{2\pi} \int_0^\pi \left(-p_\infty \cos \theta + \frac{3U\mu}{2a} \cos^2 \theta + \frac{3U\mu}{2a} \sin^2 \theta \right) a^2 \sin \theta d\theta d\phi.$$

This isn't a difficult integral to do; the term with p_∞ integrates to zero, and the rest combine to give

$$F_z = 6\pi U \mu a. \quad (6.75)$$

This is our big result – it gives the drag force on a sphere moving through a very viscous fluid, and is called *Stokes law*. Notice that the force is proportional to the velocity U as you might expect.

6.5 Stellar Structure

In this section we turn our attention to using fluid dynamics to explore the structure of *stars*. There are two pretty big things which we'll have to include in addition to the usual theory: the presence of *self-gravity*, meaning that the star is massive enough that we have to take its own gravity into account, and the presence of energy generation and transport – stars create energy through nuclear fusion at their core and transport it to the surface via mechanisms like radiative transport and convection.

6.5.1 Newtonian Gravity

We've already discussed, way back in Chapter 3, using a *gravitational potential* to describe gravity, and we'll continue that here. If \mathbf{g} is the gravitational field as usual, then we can write it in terms of a gradient of a scalar,

$$\mathbf{g} = -\nabla\Phi. \quad (6.76)$$

It'll be much easier to work with the gravitational potential rather than the field.

One difference from what we've done before, however, is the form that the field \mathbf{g} takes; after all, we're not on the surface of the Earth anymore, where $\mathbf{g} = [0, 0, -g]$, for example. Instead, it's due to the entire star itself, and we have to add up all the contributions from each bit of it. From Newton's law of gravity, the force on a mass m due to a mass m' is

$$\mathbf{F}(\mathbf{r}) = -\frac{Gmm'}{|\mathbf{r} - \mathbf{r}'|^3}(\mathbf{r} - \mathbf{r}'),$$

where \mathbf{r} is the location of mass m and \mathbf{r}' is the location of mass m' . From Newton's second law, the mass feels an acceleration

$$\mathbf{g}(\mathbf{r}) = \frac{\mathbf{F}(\mathbf{r})}{m} = -\frac{Gm'}{|\mathbf{r} - \mathbf{r}'|^3}(\mathbf{r} - \mathbf{r}')$$

(and \mathbf{g} is often called the gravitational field). At this point it's simple to generalize to get the field due to a continuous mass distribution like a star:

$$\mathbf{g}(\mathbf{r}) = -G \int_V \frac{\rho(\mathbf{r}')(\mathbf{r} - \mathbf{r}')}{|\mathbf{r} - \mathbf{r}'|^3} d\tau'. \quad (6.77)$$

This is the connection between the mass density $\rho(\mathbf{r})$ and the field $\mathbf{g}(\mathbf{r})$; what about the potential $\Phi(\mathbf{r})$? We find the potential via Poisson's equation,

$$\boxed{\nabla^2 \Phi(\mathbf{r}) = 4\pi G \rho(\mathbf{r})}. \quad (6.78)$$

This is the main equation of Newtonian gravity.

6.5.2 Energy Transport

A star generates energy at its core through fusion; stars on the so-called main sequence burn hydrogen to helium. This process is of course quite complicated, requiring understanding of the various nuclear processes that happen, but if we ignore the details, suppose that ϵ is the total energy released through fusion per unit mass per unit time – that means

$$\int_V \epsilon \rho d\tau$$

is the *rate* at which energy is generated within the star.

The star then transports that energy to the surface. There are two main processes that accomplish this: radiative transport, in which energy is carried by photons, and adiabatic convection, in which energy is carried by matter. Again, ignoring the details for now, we'll call \mathbf{F} the *flux* of energy, and the quantity

$$\int_S \mathbf{F} \cdot d\mathbf{a}$$

represents the rate that energy is transported through a surface S with area vector $d\mathbf{a}$.

In addition to these two processes, the energy within a star can change due to the work done by the pressure in the fluid and the work done by gravity. If we add up all the contributions, the total rate of energy change is

$$\frac{dE}{dt} = - \int_S p \mathbf{u} \cdot d\mathbf{a} + \int_V \mathbf{g} \cdot \mathbf{u} \rho d\tau + \int_V \epsilon \rho d\tau - \int_S \mathbf{F} \cdot d\mathbf{a} \quad (6.79)$$

(note the negative sign on the pressure and flux terms to account for the fact that the area vector $d\mathbf{a}$ points outward).

The total energy E can be written as the sum of the kinetic energy T and the internal thermal energy (per unit mass) U , so the change in energy is

$$\frac{dE}{dt} = \frac{d}{dt} \int_V \left(\frac{1}{2} \mathbf{u}^2 + U \right) \rho d\tau.$$

If we move the derivative inside the integral, it becomes the material derivative,

$$\frac{dE}{dt} = \int_V \left[\frac{D}{Dt} \left(\frac{1}{2} \mathbf{u}^2 \right) + \frac{DU}{Dt} \right] \rho d\tau.$$

We can then use the divergence theorem to turn the surface integrals in equation (6.79) into volume integrals, and, since the volume is arbitrary, drop the integrals to get

$$\rho \left[\frac{D}{Dt} \left(\frac{1}{2} \mathbf{u}^2 \right) + \frac{DU}{Dt} \right] = -\nabla \cdot (\rho \mathbf{u}) + \rho g \cdot \mathbf{u} + \rho \epsilon - \nabla \cdot \mathbf{F}. \quad (6.80)$$

One last thing: we can use Euler's equation (assuming the star is made of an ideal fluid, a good assumption) to get rid of the kinetic energy term, and the continuity equation to clean things up a bit more. Problem 6.4 has the details; equation (6.80) becomes

$$\frac{DU}{Dt} = \frac{p}{\rho^2} \frac{D\rho}{Dt} + \epsilon - \frac{1}{\rho} \nabla \cdot \mathbf{F}. \quad (6.81)$$

This equation says that the internal thermal energy of a fluid element can change in three different ways: by a change in density, by generating energy via fusion, or by transporting energy away.

6.5.3 Equations of Stellar Structure

Equations (6.78) and (6.81), along with Euler's equation and the continuity equation, constitute the theory of stellar structure. For convenience, I'll put them all together:

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho} \nabla p - \nabla \Phi \quad (\text{Euler's equation}),$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (\text{continuity equation}),$$

$$\nabla^2 \Phi = 4\pi G \rho \quad (\text{self-gravity}),$$

$$\frac{DU}{Dt} = \frac{p}{\rho^2} \frac{D\rho}{Dt} + \epsilon - \frac{1}{\rho} \nabla \cdot \mathbf{F} \quad (\text{thermal energy}).$$

Now, that's four equations, but if you count the unknowns, there are actually five: the velocity \mathbf{u} , the pressure p , the density ρ , the gravitational potential Φ , and the thermal energy U . Note that ϵ , the rate of energy generation, and F , the flux of energy, aren't strictly unknowns here; they require additional theory to compute, depending on what exactly we're modelling, so think of them as *inputs* into the theory of stellar structure. But what's the fifth equation? As discussed back in Section 5.3.1, we need an *equation of state*. More on what we'll use in a bit.

These equations are extremely complicated to solve, even numerically. But we can make some headway and produce simple but useful models of stars and other objects with a few assumptions:

- That the fluid has no time dependence – it's *steady flow*. But we'll also assume no time dependence for every variable, including energy U .
- Even stronger, we'll assume the fluid is *static* – that $\mathbf{u} = 0$. Our stars will be made up of a fluid that doesn't actually flow.
- And we'll assume spherical symmetry – there's only a dependence on the distance from the origin, r .

As you might guess, these assumptions drastically simplify our equations; they become

$$\frac{dp}{dr} = -\rho \frac{d\Phi}{dr} \quad (\text{Euler's equation}), \quad (6.82)$$

$$0 = 0 \quad (\text{continuity equation}), \quad (6.83)$$

$$\frac{1}{r^2} \frac{d}{dr} \left(r^2 \frac{d\Phi}{dr} \right) = 4\pi G\rho \quad (\text{self-gravity}), \quad (6.84)$$

$$\epsilon = \frac{1}{\rho} \frac{1}{r^2} \frac{d}{dr} (r^2 F) \quad (\text{thermal energy}). \quad (6.85)$$

But even better, we can combine the first and third equation in the following way. First, integrate the third equation for self-gravity, so that

$$r^2 \frac{d\Phi}{dr} = \int_0^r 4\pi G\rho(r')r'^2 dr'.$$

But the right hand side is (almost) just the mass contained inside a radius r :

$$M(r) = 4\pi \int_0^r \rho(r')r'^2 dr',$$

so the third equation can be written

$$\frac{d\Phi}{dr} = \frac{GM(r)}{r^2}, \quad (6.86)$$

and substituted into the first equation, giving

$$\boxed{\frac{dp}{dr} = -\frac{GM(r)\rho}{r^2}.} \quad (6.87)$$

This is the equation for *hydrostatic equilibrium*: the inward force of gravity is balanced by the outward pressure of the fluid, allowing the star to be static.

The fourth equation, for thermal energy, says that any energy produced (ϵ) within the star must be balanced by the energy flux (F) – this is a statement of *thermal equilibrium*. It's more common to write this in terms of the *luminosity* L , which is the energy emitted per unit time by a radial shell of the fluid, in which case the flux is

$$F = \frac{L}{4\pi r^2}. \quad (6.88)$$

Then the fourth equation becomes, in terms of the luminosity,

$$\boxed{\frac{dL}{dr} = 4\pi r^2 \rho \epsilon.} \quad (6.89)$$

Finally, we need to discuss the equation of state – but there are two possibilities. If we assume the pressure in the fluid is provided purely by the radiation emitted by the gas, then the equation of state is

$$p_{\text{rad}} = \frac{1}{3}aT^4, \quad (6.90)$$

where $a = 7.566 \times 10^{-16} \text{ J m}^{-3} \text{ K}^{-4}$ is called the radiation constant (it's related to the Stefan-Boltzmann constant if you're more familiar with that). There could also be a contribution to the pressure from the fluid itself. If we assume it's an ideal gas, then we have the usual (see Section 5.3.1) gas law

$$p_{\text{gas}} = \rho \kappa T. \quad (6.91)$$

In general, both of these sources of pressure are important, and we should include them both:

$$p = p_{\text{rad}} + p_{\text{gas}} = \frac{1}{3}aT^4 + \rho \kappa T. \quad (6.92)$$

Suppose that the gas pressure makes up some fraction of the total – call the fraction β , so that

$$p_{\text{gas}} = \beta p = \rho \kappa T.$$

Radiation pressure makes up the rest:

$$p_{\text{rad}} = (1 - \beta)p = \frac{1}{3}aT^4.$$

We can use these two equations to eliminate the temperature, giving the pressure in terms of the density alone. It turns out to take on the simple form

$$p = k\rho^{4/3}, \quad (6.93)$$

where the constant k is given by

$$k = \left[\frac{3(1 - \beta)\kappa^4}{a\beta^4} \right]^{1/3}.$$

Stars with an equation of state given by equation (6.93) are called *Eddington standard models*. But it turns out that the general form of the equation of state, that $p \propto \rho^\gamma$, arises in many situations – we've seen one already in Section 5.3.1 with adiabatic processes, where γ is related to the ratio of specific heats. Models which have this form are called *polytropes* and deserve a bit more discussion.

6.5.4 Polytropic Models

Suppose, then, we take as our equation of state the general form

$$p = k\rho^\gamma. \quad (6.94)$$

What does this do for us? We can use it to eliminate either pressure or density from equation (6.87). First rearrange and take the derivative of equation (6.87) to get

$$\frac{d}{dr} \left(\frac{r^2}{\rho} \frac{dp}{dr} \right) = -G \frac{dM}{dr},$$

and then use $dM/dr = 4\pi r^2 \rho$ to get

$$\frac{d}{dr} \left(\frac{r^2}{\rho} \frac{dp}{dr} \right) = -4\pi G \rho.$$

Now we can replace the pressure using equation (6.94); the derivative of it is

$$\frac{dp}{dr} = k\gamma \rho^{\gamma-1} \frac{d\rho}{dr}.$$

Plugging this into the equation above gives us an expression involving only the density:

$$\frac{k\gamma}{r^2} \frac{d}{dr} \left(r^2 \rho^{\gamma-2} \frac{d\rho}{dr} \right) = -4\pi G \rho. \quad (6.95)$$

We could in principle solve this differential equation for the density $\rho(r)$, but this is not a very simple-looking equation. We can clean it up a bit by introducing two dimensionless parameters. The first is a scaled density parameter, ϱ , such that

$$\rho(r) = \rho_c \varrho^n(r), \quad (6.96)$$

where ρ_c is a constant (with units of density) and the index n is related to the parameter γ by

$$\gamma = \frac{n+1}{n}. \quad (6.97)$$

The second parameter is a scaled radius; define

$$\eta = \frac{r}{\lambda}, \quad (6.98)$$

where λ is a constant dependent on the index n :

$$\lambda^2 = \frac{(n+1)k\rho_c^{(1-n)/n}}{4\pi G}. \quad (6.99)$$

In terms of these two new variables, equation (6.95) becomes

$$\frac{1}{\eta^2} \frac{d}{d\eta} \left(\eta^2 \frac{d\varrho}{d\eta} \right) = -\varrho^n.$$

(6.100)

This is called the *Lane-Emden equation*, and the solutions to it are polytropic models of stars.

We'll solve the Lane-Emden equation in a moment, but first we need to discuss the boundary conditions. The first is pretty simple: we want the density at the centre of the star to be *finite*, so we'll choose

$$\varrho(0) = 1. \quad (6.101)$$

This makes the constant ρ_c the value of the density at the centre of the star (hence the subscript c), something that can be fit to model a particular system. The second boundary condition needs a bit more motivation.

Consider a small volume at the centre of the star with radius δ . The volume within δ is

$$V = \frac{4}{3}\pi\delta^3,$$

and the mass enclosed by that radius is

$$M = \frac{4\pi}{3} \bar{\rho} \delta^3,$$

where $\bar{\rho}$ is the *average* density within the volume. From equation (6.87), the change in pressure is then

$$\frac{dp}{dr} = -\frac{4\pi}{3} G \bar{\rho}^2 \delta.$$

In the limit as $\delta \rightarrow 0$, then, we have that

$$\frac{dp}{dr} \rightarrow 0.$$

Now, for a polytropic model, the density is proportional to the pressure, so this also says

$$\frac{d\rho}{dr} \rightarrow 0 \quad \text{as} \quad r \rightarrow 0.$$

In terms of the dimensionless variables, our second boundary condition becomes

$$\frac{d\varrho}{d\eta} = 0 \quad \text{at} \quad \eta = 0. \tag{6.102}$$

We're ready to solve the Lane-Emden equation and produce our first models for a star. It turns out, however, that there's only a few cases for which it can be solved analytically: for values of $n = 0, 1, 5$, and $n = \infty$. I'll walk you through a couple here, and you can work on Problem 6.5 for others.

Example 6.2 – $\mathbf{n} = 0$. The easiest case is $n = 0$, in which case the Lane-Emden equation becomes

$$\frac{1}{\eta^2} \frac{d}{d\eta} \left(\eta^2 \frac{d\varrho}{d\eta} \right) = -1.$$

This case be solved by integrating twice, giving

$$\varrho(\eta) = c_1 - \frac{c_2}{\eta} - \frac{1}{6} \eta^2.$$

Fitting the boundary conditions requires $c_1 = 1$ and $c_2 = 0$, and the dimensionless density is

$$\varrho(\eta) = 1 - \frac{1}{6} \eta^2.$$

Notice that the density drops to zero at a radius $\eta_s = \sqrt{6}$ – this is the surface of the star.

But this is a strange model for star, since, according to equation (6.96), the density is actually constant ρ_c throughout. So, not a very good model of a star – but it does turn out to be a reasonable model of a rocky planet like the Earth.

Example 6.3 – $n = 5$. If $n = 5$, the Lane-Emden equation becomes

$$\frac{1}{\eta^2} \frac{d}{d\eta} \left(\eta^2 \frac{d\varrho}{d\eta} \right) = -\varrho^5.$$

This has the solution

$$\varrho(\eta) = \frac{1}{\sqrt{1 + \eta^2/3}}.$$

Unlike the previous example, where the density dropped to zero at a certain point, in this case the density becomes zero at infinity – evidently $n = 5$ models an infinitely large star! Actually, this case – usually called a *Plummer sphere* – can be a simple model of systems of stars like globular clusters or galaxies.

Neither of these two examples were particularly good model for *stars*, the objects we set out to investigate in the first place. Other values of n do a better job of that: $n = 1.5$ provides a good model of convective star cores as well as low mass white dwarfs, for example, and $n = 3$, which corresponds to the Eddington standard model, models the radiation zones of main-sequence stars as well as high mass white dwarfs. However, those cases don't have an analytic solution and the Lane-Emden equation must be solved numerically. The densities of some of these models are shown in Figure 6.14.

6.6 Computational Methods

Although we've now spent almost 200 pages talking about and solving fluid dynamics problems, it should be clear that what we've covered is actually drastically limited in scope. Here's what we've been able to solve:

- Viscous fluid flowing in a straight line or in a circle

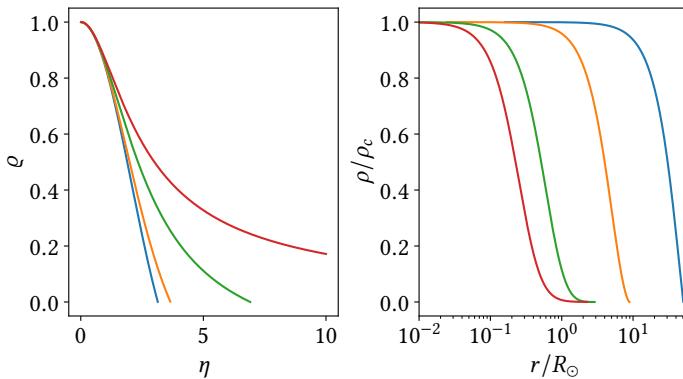


Figure 6.14: Density profiles for various polytropes. Left: the dimensionless density parameter $\rho(\eta)$. Right: the density $\rho(r)$, using solar values for various constants, plotted in terms of the central density of the sun and solar radii. Note that the horizontal axis is log scaled. Blue: $n = 1$. Orange: $n = 1.5$. Green: $n = 3$. Red: $n = 5$.

- Ideal flow past various simple objects, like circular cylinders or infinitely long aerofoils
- The motion and interaction of idealized vortices
- Small amplitude surface waves
- Small amplitude sound waves
- The free surface of a rotating fluid or a dam break
- And a few other advanced topics in this chapter: a simple boundary layer model, very viscous flow past a sphere, a simple model for a static star.

That list is both impressive and also short. Although there's more theory we could talk about – boundary layers could fill its own chapter, and we could at least touch on instability and turbulence – the fact is that most applications of fluid dynamics today are much too complicated to handle by solving equations by hand, even with the impressive set of tools we've learned here. If we want to understand the lift and drag forces on a real airplane, model weather and climate, examine high-speed plasma outflows from accretion disks surrounding black holes, or check how close our dam breaking theory matches an actual dam break (without causing one in actuality), we need to turn to computational methods of solving the Navier-Stokes and related equations.

These methods are outside the scope of this textbook, so I won't be providing too much detail, but they're worth discussing even a little because of their importance.

6.6.1 Computational Fluid Dynamics

Traditionally, computational methods involved solving the Navier-Stokes equation, along with the continuity equation and conservation of energy, by discretizing them on a grid of some sort. This process can be made computationally easier by combining all three of these equations into one:

$$\frac{\partial(\rho\phi)}{\partial t} + \nabla \cdot (\rho\phi\mathbf{u}) = \nabla \cdot (\Gamma\nabla\phi) + S_\phi, \quad (6.103)$$

where the variable³ ϕ stands for either 1 (in which case this equation reduces to the continuity equation), a component of velocity (for example, if $\phi = u$, the equation reduces to the x component of the Navier-Stokes equation), or the internal energy (in which case it becomes conservation of energy). The variable Γ , called the diffusion coefficient, is just the viscosity if ϕ is a velocity component and is the thermal conductivity if it is the energy. The last term, S_ϕ , is called the *source term*, and could represent a force or an energy source.

Turning this equation into a form suitable for solving it computationally is a complex problem, and various schemes are in use. The simplest is the *finite difference method*, in which quantities are evaluated on a grid in space. The derivatives in equation (6.103) become finite differences (hence the name); for example, the *forward difference* scheme approximates

$$\frac{df}{dx} \approx \frac{f(x + \Delta x) - f(x)}{\Delta x}.$$

Any ordinary differential equations I solved numerically in the course of this textbook were done using a finite difference method. For the Navier-Stokes equation, though, this method is not particularly flexible and isn't often used except in cases where a regular grid is appropriate – for example, simulations of weather.

The *finite volume method* is more popular for solving fluid dynamics problems. In this method, the fluid domain is divided into small *control volumes*; for each control volume – or *cell* – the volume average of the fluid variables can be evaluated. Variables can change due to a flux entering into the volume, and this must be balanced by a corresponding flux

³This ϕ is not the spherical coordinate or the velocity potential, sorry once again for overusing this letter.

out of the volume to ensure conservation of that quantity. This method requires rewriting equation (6.103) by integrating over the volume, and using the divergence theorem to reduce the volume integrals to surface integrals where there are divergences. The main equation then becomes

$$\frac{\partial}{\partial t} \left(\int_V \rho\phi \, d\tau \right) = - \int_S \rho\phi \mathbf{u} \cdot d\mathbf{a} + \int_S \Gamma \nabla \phi \cdot d\mathbf{a} + \int_V S_\phi \, d\tau,$$

which we can interpret as saying that any increase in ϕ inside the cell must be because of *convection* (first term on the right hand side) or *diffusion* (second term) or the generation of more ϕ (the third term). Finite volume methods are more flexible than finite differences, and can be used with both a regular mesh of cells or an irregular one – useful in solving problems involving complicated shapes like the surface of a car.

A third technique is the *finite element method*, in which the fluid domain is again divided up into smaller elements. However, unlike the previous two methods, each element is modelled independently assuming each variable changes according to a simple function. The models for each element are then combined into a larger system of equations that models the entire system.

If you’re interested in exploring computational fluid dynamics, I recommend the software SU2 (su2code.github.io), an open source package written in c++ and Python. SU2 can solve Euler’s equation and the Navier-Stokes equation for both incompressible and compressible fluids, and is very flexible: although designed originally for aerodynamic shape optimization, it can handle potential flow, elasticity, and flows containing electrodynamics or chemical reactions. Figure 4.23 was created using SU2 – it solved the Navier-Stokes equations on an irregular multi-level grid using a finite volume method for flow past a cylinder.

6.6.2 Smoothed Particle Hydrodynamics

A very different approach to solving the Navier-Stokes and related equations is taken by *smoothed particle hydrodynamics*, first developed to simulate polytropic models of stars. Rather than using a grid or mesh, this method uses particle-based techniques to simulate interactions between volume elements – sometimes in this case called *control masses*. The particles carry all physical properties of the system as they move; this viewpoint, in which the coordinate system follows the particles, is called *Lagrangian*. Throughout this textbook, on the other hand, I’ve used a *Eulerian* viewpoint in which the fluid properties are described at fixed points.

How exactly does this work? Well, consider for simplicity Euler's equation,

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla p + \frac{1}{\rho}\mathbf{f},$$

where \mathbf{f} represents any external forces present (for example, gravity would be ρg as usual). Because of the Lagrangian viewpoint, we can treat the material derivative as $d\mathbf{u}/dt$, where $\mathbf{u} = d\mathbf{r}/dt$. Given an initial configuration of the fluid, represented as particles at particular positions, Euler's equation tells us how to update those positions and other quantities by approximating the derivatives as finite differences. One particular particle, for example, would have its position and velocity updated according to

$$\begin{aligned}\Delta\mathbf{r} &= \mathbf{u}\Delta t, \\ \Delta\mathbf{u} &= \left(-\frac{1}{\rho}\nabla p + \frac{1}{\rho}\mathbf{f}\right)\Delta t,\end{aligned}$$

where Δt is the time step.

One issue with this scheme, however, is how to compute the derivatives that show up in Euler's equation (or the Navier-Stokes, which has a second order derivative). Consider the pressure; it can be written in terms of a *interpolating or smoothing kernel* $W(\mathbf{r})$ as

$$p(\mathbf{r}) \approx \int p(\mathbf{r}')W(\mathbf{r} - \mathbf{r}')d\tau'.$$

Note that if $W(\mathbf{r}) = \delta^3(\mathbf{r})$, then this relation is exact; most commonly the kernel is a Gaussian,

$$W(\mathbf{r}) = \left(\frac{1}{h\sqrt{\pi}}\right)^3 e^{-r^2/h^2},$$

where h is the smoothing length scale (in the limit $h \rightarrow 0$, the Gaussian becomes the Dirac delta). Now, if we want to calculate the gradient of the pressure, the ∇ operator only acts on function of the coordinate \mathbf{r} – so we can write

$$\nabla p(\mathbf{r}) = \int p(\mathbf{r}')\nabla W(\mathbf{r} - \mathbf{r}')d\tau,$$

and perform the gradient by hand.

As with traditional CFD methods, SPH must be discretized for calculation on a computer. In this case it's straightforward – the integrals

become sums in the usual way. The volume element $d\tau$ can be written in terms of the density,

$$d\tau = \frac{m}{\rho},$$

where m is the mass of the particle under consideration, and the pressure gradient takes the form

$$\nabla p \approx \sum_j m \frac{p(\mathbf{r}_j)}{\rho_j} \nabla W(\mathbf{r} - \mathbf{r}_j).$$

The sum is over all particles, and ρ_j is the density of the j th particle, which can be found from

$$\rho_i = \sum_j m_j W(\mathbf{r}_i - \mathbf{r}_j).$$

Finally, then, Euler's equation for updating the velocity of the i th particle becomes

$$\frac{\Delta \mathbf{u}_i}{\Delta t} = -\frac{1}{\rho_i} \sum_j m_j \frac{p_j}{\rho_j} \nabla W(\mathbf{r}_i - \mathbf{r}_j) + \frac{1}{\rho_i} \mathbf{f}_i \quad (6.104)$$

(actually, this is usually written in a way that's more efficient for calculation, but this should give you an idea of what's involved). If viscous forces are to be included, the Laplacian in the Navier-Stokes equation is handled in a similar manner to the gradient here.

If you're interested in exploring smoothed particle hydrodynamics, I recommend taking a look at PySPH (pysph.readthedocs.io), an open source SPH package written in Python. It can handle weakly compressible SPH for free surface flows (like the dam break shown in Figure 6.15), incompressible fluids, elastic dynamics, various boundary conditions, and more.

6.6.3 Lattice Boltzmann Method

Now for something completely different – the lattice Boltzmann method doesn't even try to solve the Navier-Stokes equation. It treats the fluid on a grid – called a lattice in this case – but limits fluid particles to only two behaviours: they can *stream* or they can *collide*. It does this by considering the collisional Boltzmann equation,

$$\frac{\partial f}{\partial t} + (\mathbf{u} \cdot \nabla) f + \nabla \Phi \cdot \frac{\partial f}{\partial \mathbf{u}} = \left(\frac{\partial f}{\partial t} \right)_{\text{coll}}, \quad (6.105)$$

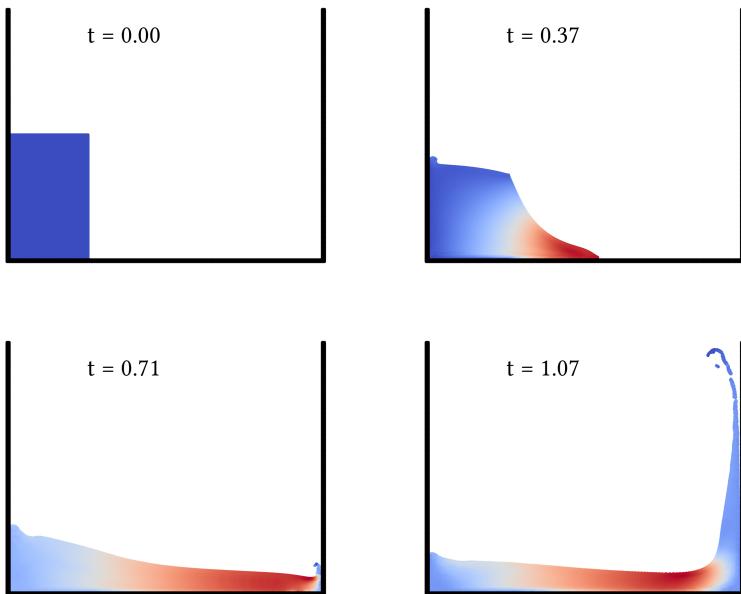


Figure 6.15: Smoothed particle hydrodynamics simulation of the two dimensional dam break problem using PySPH. The red colour indicates faster horizontal speed u .

which is an equation for the *distribution function* $f(\mathbf{r}, \mathbf{u}, t)$. This function describes the phase-space density of the fluid. This means that the quantity

$$f(\mathbf{r}, \mathbf{u}, t) d^3\mathbf{r} d^3\mathbf{u}$$

is the mass of fluid particles having positions in the volume $d^3\mathbf{r}$ centred on \mathbf{r} and with velocities in the range $d^3\mathbf{u}$ centred on \mathbf{u} . Any physical quantity can be calculated from the distribution function; for example, the mass density is

$$\rho(\mathbf{r}, t) = \int f(\mathbf{r}, \mathbf{u}, t) d^3\mathbf{u}.$$

In equation (6.105), you might recognize the material derivative of f on the left hand side. In the third term, Φ represents the potential due to some external force, if one is acting. And the term on the right hand side is there to handle interactions between fluid particles – in general, this term can be very difficult to calculate. The so-called BGK approximation

(from Bhatnagar, Gross, and Krook) models the collisions as

$$\left(\frac{\partial f}{\partial t}\right)_{\text{coll}} = -\frac{f - f_{\text{eq}}}{\tau},$$

where f_{eq} is an equilibrium state and τ is the timescale for collisions to occur. A common equilibrium state is an isothermal one, so that the collisions try to drive the fluid towards a state where the temperature is the same everywhere.

If we drop the external potential term and use the BGK model, the Boltzmann equation becomes

$$\frac{\partial f}{\partial t} + (\mathbf{u} \cdot \nabla)f = -\frac{f - f_{\text{eq}}}{\tau}.$$

The left hand side describes the movement of fluid particles and the right their collisions. This is the heart of the lattice Boltzmann method.

As with other methods, though, this equation needs to be discretized on a lattice so that the distribution function f can be evolved in time. Fluid quantities can then be found by summing over the nearby neighbours of the lattice site; for example, the density is

$$\rho = \sum_i f_i,$$

and the velocity is

$$\mathbf{u} = \frac{1}{\rho} \sum_i f_i \mathbf{u}_i.$$

This method can be simple to implement and, in cases where it's applicable, much faster than standard computational methods. There are various software packages available, but a simple open source implementation is given by Philip Mocz (github.com/pmocz). Figure 6.16 shows a simulation based on this code for fluid flowing past a cylinder; streamlines and vorticity are shown for different times. You can compare with Figure 4.23, which does the same thing but with standard CFD. The lattice Boltzmann method does a good job with vortex shedding and the formation of a vortex street.

Problems

Problem 6.1 – Pressure and the Stress Tensor. Show that, for the Newtonian stress tensor (equation 6.1.3), the pressure is the negative of the average of the three normal stresses. Explain why this makes sense.

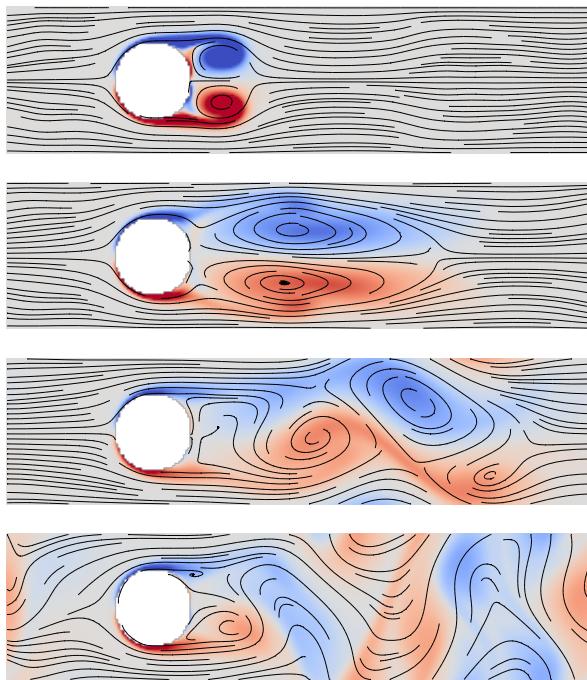


Figure 6.16: Lattice Boltzmann method simulation of fluid flowing past a cylinder, shown at four different times. Streamlines are shown, in addition to the vorticity in the fluid (with red begin positive vorticity and blue negative).

Problem 6.2 – Flow around a sphere. Consider ideal incompressible flow, uniform at infinity, around a sphere of radius a , as in Section 6.4.1.

(a) From the velocity potential $\phi(r, \theta)$ in equation 6.41, show that the velocity of the flow is

$$u_r(r, \theta) = U \cos \theta \left(1 - \frac{a^3}{r^3} \right) \quad \text{and} \quad u_\theta(r, \theta) = -U \sin \theta \left(1 + \frac{a^3}{2r^3} \right).$$

(b) Show in spherical coordinates that the Stokes stream function satisfies

$$(\mathbf{u} \cdot \nabla) \Psi = 0$$

and is thus constant on streamlines.

(c) Show that for the flow around a sphere, the Stokes stream function can be written as

$$\Psi(r, \theta) = -\frac{1}{2}U \left(r^2 - \frac{a^3}{r} \right) \cos^2 \theta$$

or

$$\Psi(r, \theta) = \frac{1}{2}U \left(r^2 - \frac{a^3}{r} \right) \sin^2 \theta.$$

Are these equivalent? Why or why not?

Problem 6.3 – Terminal velocity for a dropped sphere. Suppose you drop a steel sphere, of radius 1 cm, into a bucket of glycerine. What is the terminal speed of the sphere in the glycerine in cm/s?

Problem 6.4 – Rate of change of thermal energy.

(a) From Euler's equation (3.1), derive the *mechanical energy equation*,

$$\frac{D}{Dt} \left(\frac{1}{2} \mathbf{u}^2 \right) = -\frac{1}{\rho} \mathbf{u} \cdot \nabla p + \mathbf{u} \cdot \mathbf{g}.$$

(Hint: Start by taking the dot product of Euler's equation with \mathbf{u} .)

(b) From the continuity equation (1.18), show that you can write the divergence of the velocity as

$$\nabla \cdot \mathbf{u} = -\frac{1}{\rho} \frac{D\rho}{Dt}.$$

(c) Thus, show that equation (6.80) becomes equation (6.81).

Problem 6.5 – Polytropes. Consider a polytrope with $n = 1$.

- (a) Solve the Lane-Emden equation to find the dimensionless density $\varphi(\eta)$ and calculate the value of the surface η_s .
- (b) Show that the total mass within the star is $M = 4\pi^2 \rho_c \lambda^3$.
- (c) Define the dimensionless pressure as

$$\varphi = \frac{p}{k \rho_c^n}$$

and find $\varphi(\eta)$ for this case.

(d) Plot the dimensionless density and dimensionless pressure.