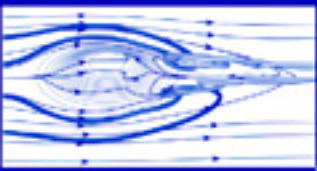
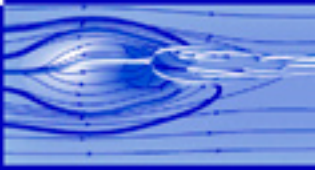


Fluid Dynamics with a Computational Perspective

Paul A. Durbin
Gorazd Medic



CAMBRIDGE

CAMBRIDGE www.cambridge.org/9780521850179

This page intentionally left blank

FLUID DYNAMICS WITH A COMPUTATIONAL PERSPECTIVE

Modern fluid dynamics is a combination of traditional methods of theory and analysis and newer methods of computation and numerical simulation. Underlying both are the principles of fluid flow. *Fluid Dynamics with a Computational Perspective* synthesizes traditional theory and modern computation. It is neither a book on methods of computation, nor a book on analysis; it is about fluid dynamics – consistent with the state of the art in that field. The book is ideal for a course on fluid dynamics. The early chapters review the laws of fluid mechanics and survey computational methodology and the subsequent chapters study flows where the Reynolds number increases from creeping flow to turbulence, followed by a thorough discussion of compressible flow and interfaces. Although all significant equations and their solutions are presented, their derivations are informal. References for detailed derivations are provided. A chapter on intermediate Reynolds number flows provides illustrative case studies by pure computation. Elsewhere, computations and theory are interwoven.

Paul A. Durbin is the Martin C. Jischke professor of aerospace engineering at Iowa State University. He was previously a professor in mechanical engineering at Stanford University. His research interests are in turbulence and transition, including computation, theory, and analytical modeling. He is a member of AIAA and ASME and a Fellow of APS. He is an associate editor of the ASME *Journal of Fluids Engineering*. He has extensive experience in teaching fluid dynamics and has written *Statistical Theory and Modeling for Turbulent Flow* and numerous articles.

Gorazd Medic is a research associate at the Center for Integrated Turbulence Simulations of the Mechanical Engineering Department at Stanford University. His research interests are in turbulence, numerical methods, and high-performance computing. He is a member of AIAA, ASME, APS, and SIAM. He has extensive experience in computational fluid dynamics for a variety of applications ranging from aircraft engines to biomechanical systems.

Fluid Dynamics with a Computational Perspective

PAUL A. DURBIN

Iowa State University

GORAZD MEDIC

Stanford University

CAMBRIDGE
UNIVERSITY PRESS

CAMBRIDGE UNIVERSITY PRESS
Cambridge, New York, Melbourne, Madrid, Cape Town, Singapore, São Paulo

Cambridge University Press
The Edinburgh Building, Cambridge CB2 8RU, UK
Published in the United States of America by Cambridge University Press, New York
www.cambridge.org
Information on this title: www.cambridge.org/9780521850179

First published in print format 2007

ISBN-13 978-0-511-35292-8 eBook (Adobe Reader)
ISBN-10 0-511-35292-1 eBook (Adobe Reader)

ISBN-13 978-0-521-85017-9 hardback
ISBN-10 0-521-85017-7 hardback

To Cinian, Seth, and Sanja, who were patient and supportive while we wrote, and to Professor Enoch Durbin, for a lifetime of encouragement

Contents

Preface *page* xi

1 Introduction to Viscous Flow **1**

1.1 Why Study Fluid Dynamics? 1
1.2 Viscosity 2
1.3 Navier–Stokes Equations 5
1.4 Reynolds Number 10
1.5 Parallel Flow Approximation 14
1.5.1 Couette–Poiseuille Flow 14
1.5.2 Oscillatory Boundary Layer 19
1.6 Some Basics 21
1.6.1 One-Dimensional Conservation Laws 22
1.6.2 Bernoulli's Equation 24
1.6.3 Unsteady Flow 29
1.7 Swirl 31
1.7.1 Computation of Swirling Flow in an Expansion 34
1.8 Vorticity and Its Dynamics 35
1.8.1 Vortex Kinematics and Dynamics 37
1.8.2 Stretching and Rotation 42
1.8.3 Origin of Vorticity 44
1.8.4 Circulation and Lift 46
1.8.5 Diffusion of Vorticity 49
1.8.6 Visualization of Vortices 52
1.9 Turbulence 55

2 Elements of Computational Analysis **61**

2.1 Geometry and Grid Generation 63
2.1.1 Computational Domain 63
2.1.2 Discrete Representation of the Geometry 67
2.1.3 Structured Grids 68
2.1.4 Unstructured Meshes 72
2.1.5 Mesh Quality 74
2.2 Computation of Fluid Flow 78
2.2.1 Discrete Equations 79
2.2.2 Centered and Upwind Differencing 82

2.2.3 Solvers 86
2.2.4 Boundary Conditions 90
2.3 Solution (In-)Accuracy 93
2.3.1 Residuals and Convergence 93
2.3.2 Grid Independence 94
2.4 Postprocessing and Visualization 97
2.4.1 Streamlines 98
2.4.2 Vectors and Contours 100
2.4.3 Quantitative Data 102
2.5 Packages and Codes 103

3 Creeping Flow **108**

3.1 The Low Reynolds Number Limit 109
3.1.1 Stokes Flow around a Sphere 111
3.1.2 Resistance Matrix 116
3.1.3 Resistance Due to Relative Motion 119
3.1.4 Computation of Resistance Coefficients 120
3.1.5 Eddies in Stokes Flow 122
3.2 Hydrodynamic Lubrication 124

4 Intermediate Reynolds Numbers **132**

4.1 Separation, Vorticity, and Vortex Shedding 132
4.1.1 Five Examples 133
4.1.2 Secondary Circulation 151
4.2 Wakes, Jets, and Mixing Layers 155
4.2.1 Scaling Shear Layer Evolution 156
4.2.2 Entrainment 159
4.2.3 Free and Impinging Round Jets 160

5 High Reynolds Number Flow and Boundary Layers **167**

5.1 Formal Definition of the Boundary Layer 168
5.1.1 Growth of the Boundary Layer 172
5.1.2 Pressure Gradients and Separation 175
5.1.3 Gridding 178
5.2 Approximate Equations 181
5.3 Three-Dimensional Boundary Layers 183
5.4 Separation in Three Dimensions and Critical Points 185
5.5 High Reynolds Number Flow over a Spheroid 188
5.6 Potential Flow 191
5.6.1 Point Sources 192
5.6.2 Images 196
5.6.3 Point Vortices and Circulation 197
5.6.4 Stream Function 199
5.6.5 Added Mass 201

6 Turbulent Flow 210
6.1 Computational Approaches 211
6.2 Statistically Averaged Method 211
6.2.1 Eddy Viscosity Model 215
6.2.2 Computed Example 220
6.2.3 Limitations of Scalar Eddy Viscosity Models 222
6.3 Turbulent Shear Flows 225
6.3.1 Boundary Layers 225
6.3.2 Wall Functions 233
6.3.3 Free Shear Layers 236
6.3.4 Computed Axisymmetric Jets 237
6.4 Eddy Simulation 239
6.4.1 Spectrum of Turbulence 239
6.4.2 Direct Numerical Simulation 242
6.4.3 DNS of Isotropic Turbulence 244
6.4.4 Approximate Simulation of Large Eddies 246
6.5 Instability Theory 254
6.5.1 Inflection Point Theorem 256
6.5.2 Instability in Boundary Layers 259

7 Compressible Flow 264
7.1 Thermodynamics 264
7.1.1 First Law 265
7.1.2 Equation of State 266
7.1.3 Entropy and Irreversibility 266
7.1.4 First Law with Flow 268
7.1.5 Isentropic Relations 270
7.2 Mach Waves 271
7.2.1 Speed of Sound 272
7.2.2 Occurrence of Shock Waves 273
7.3 One-Dimensional Gas Dynamics 276
7.3.1 Normal Shock 276
7.3.2 Supersonic Flow over a Hemisphere 278
7.3.3 Expansion Fan 281
7.3.4 Supersonic Flow over a Wedge 284
7.4 Quasi-One-Dimensional Gas Dynamics 286
7.4.1 Shock Patterns and Shock-Induced Separation 292
7.5 Computation of Compressible Flows 300

8 Interfaces 307
8.1 Interface Conditions 308
8.2 Surface Tension 311
8.2.1 Capillarity and Contact Angle 314
8.2.2 Disintegration by Capillarity 316
8.2.3 Simulation of Capillary Breakup 320

8.3 Inviscid Free Surface 321
8.3.1 Group Velocity and Its Connection to Surface Excrescence 323
8.3.2 Sloshing 328
8.3.3 Numerical Simulation of Sloshing 332
8.4 Volume of Fluid Method 334
8.4.1 Pouring a Viscous Liquid 336

Bibliography 343

Index 345

Preface

This is a book on fluid dynamics. It is not a book on computation. Many excellent books on fluid dynamics are available: why is another needed?

In recent decades, numerical algorithms and computer power have advanced to the point that computer simulations of the Navier–Stokes equations have become routine. This vastly expands our ability to solve these equations, further extending our understanding of fluid flow and providing a tool for engineering analysis. Computer simulations are solutions of a different nature from classical exact and approximate solutions. They are numerical data rather than formulas. One of our objectives in this text is to relate computer solutions to theoretical fluid dynamics. Indeed, it is this goal, rather than computation as a tool for complex engineering analysis, that provides the guideline for this text. Computer solutions can reproduce closed-form and approximate solutions; they can illuminate the merits and limits of simple analyses; and they can provide entirely new solutions of varying degrees of complexity. The time is ripe to integrate computer solutions into fluid dynamics education.

From a pedagogical perspective, readily available, commercial computational fluid dynamics (CFD) software provides a new resource for teaching fluid dynamics. This software converts CFD from a technique used by researchers and engineers in industry into a readily accessible facility. It is a challenge to integrate such software packages into the educational structure. Most of the examples in this book have been computed with commercial software, and exercises to be solved with such software have been suggested. How far to go in this direction was a true quandary. We started out ambitiously, intending an intimate use of commercial tools, but backed off, deciding on a text that provides the reader with illustrative computations. Grids and specifications needed to effect computer simulation are described, but far short of the level of detail found in tutorials. The endeavor was to make the book useable either on its own or as an adjunct to an integrated course on fluid dynamics and computation. A lecturer might supplement this text with a computer laboratory, instructing students via tutorials.

This certainly is not a book on methods of computational fluid dynamics. We assume that a student who uses this text in conjunction with computational analysis has access to CFD software, including both flow solver and mesh generator. Methods and algorithms are mentioned only to the extent that they bear on the fidelity of computations. In other words, the perspective is that of a code user, not of a

developer. One impetus for this book was the observation by many of our colleagues that students are increasingly proficient at using software, but often without the understanding of fluid dynamics needed to use it effectively. We hope this text will be a resource for educating prospective software users.

Despite the power of computer simulation, it has not supplanted theory and analysis in fluid dynamics teaching, even at advanced levels; nor have theoretical efforts diminished within the research community. No wholesale displacement of theory by computation is remotely on the horizon. The complexity and intricacy of flow phenomena are too great, and the range of applications are too vast, for any single approach to be sufficient. Hence, in this book, we endeavor both to present some basics of the theory of fluid flow and to explore them by computer simulation. The intent is a marriage between classical theory and modern computer-aided analysis.

Deciding how much mathematics to include was another quandary. A great many texts detailing analyses and solutions to governing equations are available. It is not our intent to reproduce such material. At the same time, there is a need to establish a framework for computer-aided analysis. This means that equations and solutions must be cited. To a large extent, we have quoted results with informal derivations, providing references where detailed developments can be found. In some cases, formulas might appear out of the blue, but in almost all cases, some basis has been provided, without rigor.

The prerequisite to this book is a course on basic fluid dynamics, including elementary viscous flow. The book is self-contained, but the pace might seem fast without the prerequisite. It is directed to students at advanced undergraduate level or to graduate students at master's level. It is also meant for scientists and engineers who want background in viscous flow phenomena.

After two chapters that provide background on the laws of fluid mechanics and survey computational methodology, the next four chapters increase the Reynolds number from creeping flow to turbulence. They are followed by a chapter on compressible flow and a final chapter on interfaces. We have been guided by the content of standard curricula in fluid mechanics.

Acknowledgments

We are extremely grateful to Frank Albina, George Constantinescu, Gianluca Iaccarino, Tomoaki Ikeda, Georgi Kalitzin, Yang Liu, Milovan Peric, Jorg Schluter, Meng Wang, Edwin van der Weide, and Xiaohua Wu for providing figures, as well as to the Institute for Mathematical Sciences, Singapore National University, for its hospitality during part of the writing. Peter Gordon of Cambridge University Press has been wonderfully encouraging.

1 Introduction to Viscous Flow

1.1 Why Study Fluid Dynamics?

Fluid dynamics is a branch of classical physics. It is an instance of continuum mechanics. A fluid is a continuous, deformable material. It is a material that flows in response to imposed forces. This is embodied in the everyday experience of draining water from a sink. The water flows under the action of gravity. It does not have a fixed shape; it fills the sink, conforming to its shape. The water flows with variable velocity, depending on its distance from the drain. All these distinguish fluid motion from solid dynamics. As another example, a pump propels water through a pipe or through the cooling system of a car. How does the reciprocating movement of the pump produce directed flow, extending to distant parts of the cooling circuit? One way or the other, the pump must be exerting forces on the fluid; one way or the other, these forces are communicated to distant portions of the fluid and sets them in motion. It is far from obvious what the nature of that flow will be, especially in a complex geometry. It may be laminar, it may be turbulent; it may be unidirectional, it may be recirculating.

Recirculation is the occurrence of backflow, opposite to the direction of the primary stream. This can be seen behind the pedestals supporting a bridge in a swift river. Despite the strong current, the flow direction reverses, and a circulating eddy forms in a region behind the pedestal. How is such behavior understood and predicted? An understanding requires knowledge of viscous action, of vorticity, of turbulence, and of the governing equations.

As a fluid flows around an obstruction, different points in the fluid flow with different velocities. The motion is continuous; hence, the velocity varies smoothly with position. The fluid might be imagined to be divided into infinitesimal volumes – which can be referred to as fluid particles. Then the fluid flow involves relative movement of the particles. Because the flow varies continuously with position, these particles must influence each other to maintain the smoothness of the flow field. That influence is via the forces of pressure and of viscous friction. It is a scientific triumph that when equations are devised to describe these forces acting on the fluid particles, the rich variety of fluid mechanics emerges. That success is most evident in computer simulation of fluid flow.

The trajectories of fluid particles are the streamlines of motion. These streamlines can be curves stretching from inflow to outflow, or they can be closed curves,

corresponding to eddying flow. The flow over a rock in a stream or past a spoon stirred through a cup of coffee are examples containing eddying flow. It is not difficult to observe phenomenology of fluid motion: hold a lighted candle, or tissue paper, at arm's length and puff very quickly. A vortex ring travels from your mouth to the candle. There is a time delay before it flickers; that is because the vortex travels with a finite speed. Now suck air into your mouth; the candle shows no evidence of flow. The vortex flows into your mouth. The flow outside is then quite different from when you blew the air out.

Some interesting and challenging scientific and engineering questions already suggest themselves. What causes the eddies and vortices? Why is the outflow so different from the inflow? What drives the flow through a conduit or over an obstacle? How is motion communicated to distant portions of the fluid? How can the flow rate be predicted? How are the streamlines of a fluid flow determined? In any but the simplest cases, these are challenging questions. There is a body of knowledge that can be called on; but one is rapidly struck with how difficult it is to answer even fairly simple questions about fluid flow.

Computer simulation has changed this, rather substantially. Flow in complex geometry can be solved numerically. The phenomena seen in laboratory experiments, and in more casual experience, can be reproduced with quantitative accuracy. To an extent, older theories and analyses have been enlarged by computation. The laws governing fluid flow have been found to be extremely accurate; one wants only sufficient computer power and efficient algorithms to produce solutions. Computer simulation becomes a method of solution, complementary to paper-and-pencil analysis. It provides further understanding of fluid flow and a tool for engineering analysis. However, computer simulations are solutions of a different nature from classical, exact, or approximate solutions. They are numerical data rather than formulas. Traditional theory is not displaced; its role evolves and it provides the understanding needed to formulate and make sense of computer-aided analysis. The additional understanding of fluid dynamics that stems from simulation should be developed in concert with theory. The motive to study fluid dynamics is to understand its phenomena. The approach is to devise and solve the laws of fluid motion.

1.2 Viscosity

At the root is the governing laws. These are the equations of conservation; conservation of mass and momentum, at the present stage. Friction is an important element of fluid flow. In the absence of friction, a flat plate, dragged tangentially across the surface of a tank of water, would slide freely and induce no movement in the water. But, in the presence of friction, motion is communicated to the water adjacent to the plate and thence a circulation is established in the tank.

Friction internal to a fluid flow is characterized by viscosity. The viscosity coefficient, μ, is an empirical property of the fluid. For instance, a liquid is more viscous than a gas. As temperature increases, liquids flow more easily (think of tar, for example) so viscosity decreases with temperature. Gases have the opposite property,

Table 1.1. *Viscosities at room temperature* (20°C)

Fluid	μ g/cm · s	ρ g/cm^3	ν cm^2/s
Air	1.8×10^{-4}	1.2×10^{-3}	0.15
Water	0.01	1.0	0.01
CO_2	1.37×10^{-4}	1.79×10^{-3}	0.077
Engine oil	10	0.89	11.2
Glycerin	7.99	1.26	6.34
Kerosene	0.024	0.78	0.031
Methyl alcohol	0.0055	0.785	0.007

that viscosity increases with temperature. That behavior is less intuitive. It originates in the increased molecular agitation as temperature increases. Clearly, μ must be measured as a function of the fluid and of the temperature. In gases, the coefficient increases approximately as the square root of temperature; in liquids, it falls as the exponential of one over temperature [$\exp(E/kT)$]. Detailed formulas need not be discussed here. Values of μ for many fluids are available in computational fluid dynamics (CFD) codes and in handbooks. The magnitude of the viscosity is essential to determining flow regimes. Table 1.1 contains a few representative values at room temperature.

The coefficient, μ, is the *dynamic* viscosity. For future reference, the *kinematic viscosity* is defined as dynamic viscosity divided by density as follows:

$$\nu = \mu/\rho. \tag{1.1}$$

Kinematic viscosity has dimensions of $length^2/time$; dynamic viscosity has dimensions of density times this or $mass/length \cdot time$. Kinematic viscosity is most relevant to constant density, incompressible flow. Oddly enough, the kinematic viscosity of liquids is often lower than that of gasses. For instance, air at 20°C and 1 atmosphere, has a kinematic viscosity of 0.15 cm^2/s; for water it is 0.01 cm^2/s. This is a consequence of the higher density of water.

Viscosity produces forces as a consequence of the relative motion of fluid particles. That might be thought of as friction associated with the particles rubbing across one another. A more correct statement is that viscous *stress* is a consequence of the fluid *rate of strain*.

The need to distinguish rate of strain from simply relative motion is because a fluid in solid body rotation experiences no viscous stress. At a macroscopic level that is clear: if the entire fluid is in solid body rotation, then in a frame rotating with the fluid there is no motion and hence no viscous stress. In a fixed frame, solid body motion means that the velocity is Ωr in the angular direction. There is relative motion in the sense that fluid at $r = 0$ is at rest, whereas at $r > 0$ it is in motion, but there is no viscous stress.

The same concept applies, less obviously, at any point in a nonrotating fluid. Relative motion can be separated into rotation and rate of strain; only the latter

produces viscous stress. The velocity is a field: at any point $\boldsymbol{x} = (x, y, z)$, three components of velocity (u, v, w) can be measured. Rate of strain is a measure of how this velocity varies from point to point within the vicinity of $\boldsymbol{x}$. Mathematically, if two points are separated by a distance $d\boldsymbol{x}$, their relative velocity is

$$u_i(\boldsymbol{x} + d\boldsymbol{x}) - u_i(\boldsymbol{x}) \approx d\boldsymbol{x} \cdot \nabla u_i,$$

where $i = 1$, 2, or 3 corresponding to u, v, or w. The last term expands to

$$dx\frac{\partial u_i}{\partial x} + dy\frac{\partial u_i}{\partial y} + dz\frac{\partial u_i}{\partial z}. \tag{1.2}$$

The convention of summation on repeated subscripts permits this to be written equivalently as

$$dx_j\frac{\partial u_i}{\partial x_j}.$$

Because the same index, j, appears twice in this product, the convention is that j is summed from 1 through 3, so that this is exactly the same expression as Eq. (1.2). This is a rather terse introduction to index notation. The uninitiated reader might want to write out corresponding formulas in index notation and in Cartesian components. That exercise is illustrated in the next paragraph.

We now introduce the separation of the velocity gradient into rate of strain and rotation; it is equivalent to a separation into symmetric and antisymmetric components, respectively. Specifically,

$$\frac{\partial u_i}{\partial x_j} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right). \tag{1.3}$$

The first term on the right is the rate of strain, which can be denoted S_{ij}; the second is minus the rate of rotation, which can be denoted $-\Omega_{ij}$:

$$\begin{aligned} S_{ij} &= \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right), \\ -\Omega_{ij} &= \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right). \end{aligned} \tag{1.4}$$

Only S_{ij} produces viscous stress. In the standard index notation used here, i and j are dummy subscripts, for which any of the numbers $(1, 2, 3)$ corresponding to the directions (x, y, z) can be substituted. For instance, with $i = 1$ and $j = 2$, Eq. (1.3) says

$$\frac{\partial u}{\partial y} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) + \frac{1}{2}\left(\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x}\right).$$

In solid body rotation, $u = -\Omega y$ and $v = \Omega x$. The first term vanishes and the second equals $-\Omega$; the rate of strain is zero under solid body rotation. An irrotational flow is one for which the second term vanishes: $\partial u/\partial y = \partial v/\partial x$. For instance, $u = \alpha y$, $v = \alpha x$ is an irrotational straining field.

The other element in the description of viscous forces is that they are characterized as a *stress* rather than as a *force* per se. This should not be unfamiliar: pressure also is a stress. That is, it is a force per unit area, acting in the direction normal to a surface. Viscous stress is similar: it is a force per unit area, but it need not act normal to the surface; it can have both normal and tangential components. If the fluid motion is just a shearing tangential to a surface, then it is clear that the viscous stress will cause a force in the tangential direction. It is less obvious, but true, that if the fluid motion is toward the surface there will also be a component of force in the normal direction. That is best described mathematically, as will be done in the next section.

Given the stress, the corresponding force on an object is obtained by integrating it over the entire area of the surface. This can be accomplished inside a CFD code, so one need only understand the origin of the force that is being computed. The reason that the force originates as a stress is that fluids are deformable material, so their dynamical properties must be defined for the fluid particles. Stress is the force per area acting on a fluid particle. It is independent of the size of the infinitesimal fluid particle. The force, by contrast, is proportional to the size. In other words, $force = stress \cdot area$. As the area becomes tiny, the stress remains finite and the force becomes tiny. Just like pressure, viscous stress is the quantity that is defined pointwise throughout the fluid. Unlike pressure, it exists only in a flowing fluid.

Viscous stress is incorporated into the governing, Navier–Stokes, momentum equations that are solved by CFD software. The gist of those equations is the next topic.

1.3 Navier–Stokes Equations

It is assumed that the reader has studied elementary fluid mechanics and has been exposed to the basic notions of fluid flow. These include the role of pressure in momentum transport, conservation of mass in an incompressible, deformable fluid medium, and the origin of viscous, frictional forces. The last have just been discussed. This section provides an informal description of the Navier–Stokes momentum equation for constant density, incompressible flow. Because we rely on CFD software for solutions to the equations, we will abbreviate the treatment that can be found more fully in standard texts on viscous flow (White, 1991).

The Navier–Stokes equations were named for the French engineer and scientist Claude Louis Marie Henri Navier and the English mathematical physicist George Gabriel Stokes. The essential form of these equations was set forth by Navier in 1822; however, he did not properly treat the origin of viscous stress. The latter was addressed by others, in particular Poisson and Saint-Venant, but independently developed by Stokes in 1845. Stokes constructed a number of solutions to the equations of viscous flow, which confirmed their ability to describe fluid dynamical phenomena. An example is creeping flow, also called “Stokes flow,” which we discuss in

Chapter 3. Navier is properly credited for the seminal formulation of the Navier–Stokes equations and Stokes for ushering their entry into theoretical physics.

Essentially, the Navier–Stokes momentum equation is an expression of Newton's law $m\boldsymbol{a} = \boldsymbol{F}$ applied to an infinitesimal fluid volume. Here we use the convention that bold letters denote vectors. On a volumetric basis, the mass becomes mass per unit volume or density ρ. The acceleration becomes that following the fluid element, or the convective derivative of velocity, $D\boldsymbol{u}/Dt$, and the force per unit volume includes both pressure and viscous contributions, as follows:

$$\rho \frac{D\boldsymbol{u}}{Dt} = \boldsymbol{\mathcal{F}}_{\text{press}} + \boldsymbol{\mathcal{F}}_{\text{viscous}}. \tag{1.5}$$

We must flesh out the meaning of these various terms.

The equations of motion referred to fluid particles is called the *Lagrangian* description. The fluid particle occupies a position $\boldsymbol{X}(t;\boldsymbol{x}_0)$ that changes with time. The particle is labeled here by its initial location $\boldsymbol{x}_0$. It is more convenient to describe the flow in terms of the velocity at fixed points. We think of a flow field, $\boldsymbol{u}(\boldsymbol{x})$, rather than of the dynamics of particles. The only complication in applying Newton's law to the field is transforming the acceleration of the fluid particle into velocity changes at a fixed position.

To derive the requisite expression, first consider a material that is carried with the fluid element. The material has a concentration c. An observer at a fixed point, $\boldsymbol{x}$, will see the concentration change as different particles arrive. At any given time, the concentration is that of the particle currently at $\boldsymbol{x}$, that is, of the particle with $\boldsymbol{X}(t) = \boldsymbol{x}$. At time δt later, a particle that was at $\boldsymbol{X} - \delta\boldsymbol{X}$, say, will have moved to $\boldsymbol{x}$. The observer then sees its concentration $c(\boldsymbol{X} - \delta\boldsymbol{X})$. Thus the observer sees the change

$$\frac{\partial c}{\partial t} = \frac{c(\boldsymbol{X} - \delta\boldsymbol{X}) - c(\boldsymbol{X})}{\delta t} \approx \frac{-\delta\boldsymbol{X} \cdot \nabla c}{\delta t}$$

as the fluid element occupying position $\boldsymbol{x}$ changes from that at time t to that at time $t + \delta t$. $\delta\boldsymbol{X}/\delta t$ is the velocity $\boldsymbol{u}$. Hence, the motion of fluid elements produces the time variation

$$\frac{\partial c}{\partial t} = -\boldsymbol{u} \cdot \nabla c.$$

There is nothing special about concentration: the same result applies to any quantity convected with the flow. Putting both terms on the same side of the last equation shows that a transported quantity satisfies

$$\frac{Dc}{Dt} \equiv \frac{\partial c}{\partial t} + \boldsymbol{u} \cdot \nabla c = 0.$$

This is a statement that the changes at a fixed position are simply due to different elements arriving at that position, carrying their particular concentration. If the quantity were not simply convected but also underwent some change, then the right side would be nonzero.

Thus, we arrive at the expression of Newton's law at a fixed point – which is called the *Eulerian* description. The quantity being carried is now the fluid momentum, $\rho \boldsymbol{u}$. It is carried with the particles but also changes as a consequence of forces. The flow field obeys

$$\rho \left[\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} \right] = \boldsymbol{\mathcal{F}}_{\text{press}} + \boldsymbol{\mathcal{F}}_{\text{viscous}}. \tag{1.6}$$

A more general concept than the right side of Eqs. (1.5) or (1.6) is to combine pressure and viscous terms into a single stress tensor. Stresses are forces acting on a surface per unit area. *Pressure* times *area* is a force acting perpendicularly to the surface; in other words, it is a *normal stress*. Viscosity produces both *normal* and *tangential*, or shearing, stresses. The tangential stress is quite intuitive: it is analogous to the force felt when rubbing one hand over the other. As mentioned in the last section, it is also the case that viscosity produces a component of normal force, parallel to pressure. On any surface, the aggregate stress is a vector, with components both normal and tangential to the surface, composed of contributions from pressure and from viscosity.

How are forces produced by stress represented? The aggregate force can be denoted $\boldsymbol{F_s}$. It is the force produced by a stress acting on a surface. Consider that surface to be a small, differential area, dA. Further, let that area be the magnitude of a vector $d\boldsymbol{A}$ that is directed normal to the surface. The stress, $\boldsymbol{\sigma}$, now can be defined: the force is the dot product of the stress with the area vector

$$\boldsymbol{F_s} = \boldsymbol{\sigma} \cdot d\boldsymbol{A}. \tag{1.7}$$

This simply defines the *stress tensor* $\boldsymbol{\sigma}$ as a matrix relating the force vector to the area vector. Purely as a matter of consistency, Eq. (1.7) shows that stress has dimensions of force per area. In component form, the matrix relation [Eq. (1.7)] between stress and surface force is stated as

$$\begin{pmatrix} F_x \\ F_y \\ F_z \end{pmatrix} = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix} \cdot \begin{pmatrix} dA_x \\ dA_y \\ dA_z \end{pmatrix}. \tag{1.8}$$

The term stress *tensor* was slipped into the text, above. A tensor is a generalization of a vector. A vector has a direction. A tensor has one or more directions. A vector is a first-order tensor, having a single direction. Stress is a second-order tensor; it is associated with two directions. The two directions are that of the force vector and that of the area vector. Matrix $\boldsymbol{\sigma}$ relates the direction of a force acting on a surface to the area vector of that surface.

Equation (1.7), integrated over a solid surface, gives the net force exerted by the flowing fluid. When, in later chapters, we consider examples of fluid forces on objects, the quantity

$$\int \boldsymbol{\sigma} \cdot d\boldsymbol{A} \tag{1.9}$$

is being evaluated by integration over the entire surface in question.

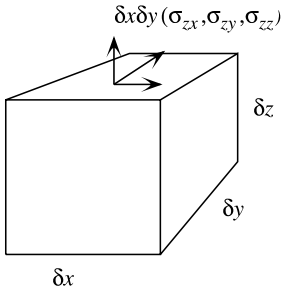


Figure 1.1. Stresses on a face of a fluid element.

The force evaluation usually can be obtained with the CFD software; but it should be understood that the computer code is providing this surface integral of the stress tensor. In fact the force can be broken down into contributions from pressure and viscosity. The accuracy of the numerical evaluations depends on how finely the mesh covers the surface and on how accurately the viscous and pressure stresses are computed by the flow solution. The force evaluation is a postprocessing of that solution.

The presence of two contributions to stress can be acknowledged by writing

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}_{\text{viscous}} + \boldsymbol{\sigma}_{\text{pressure}}, \tag{1.10}$$

Precise expressions for these two contributions will be given shortly.

Forces due to stress also act inside the fluid. Within the fluid, they are the force on the face of an infinitesimal fluid element (Figure 1.1). That element will move if there is an imbalance of forces. Consider two opposite sides of an element having oppositely directed normals. The force imbalance is due to a difference between the stresses acting on the opposite sides, that is, to a differential of stress. If $\sigma_L A$ is the force on the left side of the fluid element and $\sigma_R A$ is the force on its right side, then $(\sigma_R - \sigma_L)A$ is the force imbalance. If ℓ is the length of the element and $\mathcal{V} = A\ell$ its volume, then the resultant force is

$$\frac{\sigma_R - \sigma_L}{\ell} A\ell = \frac{\sigma_R - \sigma_L}{\ell}\mathcal{V} \approx \frac{d\sigma}{d\ell}\mathcal{V}.$$

That is, the force per unit volume is the directional derivative of the stress. The equation of motion is now $\mathcal{M}\, Du/Dt = \mathcal{V}\, d\sigma/d\ell$. The ratio $\mathcal{M}/\mathcal{V}$ is the density ρ. It takes only a bit of elaboration to recognize that the directional derivative $d\sigma/d\ell$ should be generalized to the divergence of the stress $\nabla \cdot \boldsymbol{\sigma}$. Essentially, the gradient operator gives the directional derivative.

Hence, the force that appears in the Navier–Stokes momentum equation is the divergence of stress, and Newton's law becomes

$$\rho \frac{D\boldsymbol{u}}{Dt} = \rho \left[\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} \right] = \nabla \cdot \boldsymbol{\sigma}. \tag{1.11}$$

This is simply Newton's law (1.5) when the force is caused by a stress gradient. As explained above, $\partial \boldsymbol{u}/\partial t + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u}$ is the Eulerian form for the acceleration $D\boldsymbol{u}/Dt$.

Now to make explicit the observation that stress is composed of a part due to pressure and a part due to viscosity. The part due to pressure is just pressure times the identity matrix, with a minus sign:

$$\boldsymbol{\sigma}_{\text{press}} = -p\boldsymbol{I}. \tag{1.12}$$

The minus sign arises because pressure applied to a surface acts inward to the surface. For instance, if the surface is the xy plane, an imposed high pressure will push down in the $-z$ direction on the surface. That is a special case of the general formula

$$\boldsymbol{F}_{\text{press}} = \boldsymbol{\sigma}_{\text{press}} \cdot d\boldsymbol{A} = -p\boldsymbol{I} \cdot d\boldsymbol{A} = -pd\boldsymbol{A}.$$

Pressure times *area* is the inward force. The pressure contribution to stress gives rise to a pressure gradient on the right side of Eq. (1.11): $\nabla \cdot \boldsymbol{\sigma}_{\text{press}} = -\nabla p$. This might be comforting to the reader who is wondering why there is no pressure gradient in Eq. (1.11).

The representation of viscous stress is less obvious. In a Newtonian fluid, it is assumed proportional to the rate of strain of the fluid motion, as described in §1.2. Viscosity is simply the coefficient of proportionality. This is stated as

$$\boldsymbol{\sigma}_{\text{viscous}} = \mu(\nabla \boldsymbol{u} + {}^t[\nabla \boldsymbol{u}]), \tag{1.13}$$

where $\nabla \boldsymbol{u}$ is a matrix of velocity derivatives and ${}^t[\nabla \boldsymbol{u}]$ is its transpose. Equation (1.13) is a statement in vector form of the relation

$$\sigma_{ij} = 2\mu S_{ij}$$

in index form, with S_{ij} given by Eq. (1.4). When this formula is used for an incompressible fluid, the viscous force simplifies to the Laplacian of velocity: $\nabla \cdot \boldsymbol{\sigma}_{\text{viscous}} = \mu \nabla^2 \boldsymbol{u}$. The Navier–Stokes equations of an incompressible (Newtonian) fluid assume the form

$$\rho \left[\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} \right] = -\nabla p + \mu \nabla^2 \boldsymbol{u}. \tag{1.14}$$

The pressure and viscous forces are stated explicitly here. Lengthy discussions of the derivation of this equation, and some caveats that must be made in our derivation, can be found in standard texts (Panton, 1997; White, 1991). In fact, it is an equation that is quite remarkable for its ability to describe the phenomena of fluid flow. It is equally remarkable for its mathematical intransigence. Basically, it is the momentum equation that is solved by CFD software.

That is not quite correct: (1.14) is an equation for the velocity vector $\boldsymbol{u}$ or for the u, v, and w components of velocity. It contains the further fluid properties ρ and p. Consider constant density flow, such as air at low speed or water without significant contaminant concentrations. Then pressure is the only additional dynamical variable. Another equation is needed to predict it.

In many situations, the interest is in essentially incompressible flow. Incompressible means that pressure changes produce negligible density changes: $d\rho/dp \approx 0$. The

reader might recognize that $d\rho/dp$ is one over the squared sound speed, c^{-2}, in a gas. In that case, the approximation of incompressibility is justified if the Mach number, $M = u/c$, is small. In other words, the smallness of $d\rho/dp$ is a relative statement. It says that pressure variations go primarily into accelerating the flow rather than into changing the density. Certainly air is compressible; it can be compressed in a pump, or into a tire. But for the purpose of fluid dynamics, it can be treated as incompressible if the flowing fluid does not cause significant compression. This is normally the case when the Mach number is low. Liquids are almost always incompressible. Density variations can occur, such as those due to salt dissolved in water, but, again, it is not the fluid velocity that causes density to vary.

The condition of incompressibility is that infinitesimal fluid elements retain their volume. Their shape will deform, but the net volume associated with an element is constant. We are defining the fluid element as a fixed amount of mass, so that constant density is equivalent to constant volume. The condition of incompressibility requires the divergence of the velocity to vanish:

$$\nabla \cdot u = 0 \tag{1.15}$$

or $\partial_x u + \partial_y v + \partial_z w = 0$. Essentially, this is saying that the volume deformations in the x, y, and z directions sum to zero. It may be justified as follows.

Consider a rectangular material element – for instruction, we work in two dimensions. The corners of the rectangle move with the fluid. Let the lower-left and upper-right corners be (X, Y) and $(X + \delta X, Y + \delta Y)$. The area is $\mathcal{A} = \delta X \delta Y$. The rectangle will deform as the fluid flows, but if it is incompressible, the area does not change. Then Eq. (1.15) follows from differentiating the area with respect to time and setting it to 0:

$$\begin{aligned}\frac{d\mathcal{A}}{dt} = 0 &= \delta Y \frac{d}{dt}\delta X + \delta X \frac{d}{dt}\delta Y \\ &= \mathcal{A}\left(\frac{1}{\delta X}\frac{d}{dt}\delta X + \frac{1}{\delta Y}\frac{d}{dt}\delta Y\right) \\ &= \mathcal{A}\left(\frac{\delta u}{\delta X} + \frac{\delta v}{\delta Y}\right) = \mathcal{A}\nabla \cdot \boldsymbol{u},\end{aligned}$$

where $\delta u = d\delta X/dt$ and $\delta v = d\delta Y/dt$ were substituted. In three dimensions, the same argument is applied to a fluid volume.

1.4 Reynolds Number

The boundary condition on rigid surfaces is that the fluid immediately adjacent to the surface moves with the wall velocity. This is the no-slip condition. It says that there is no discontinuity in velocity at the wall. As a stationary wall is approached, the fluid velocity tends continuously to zero. This no-slip boundary condition is not indisputable. In rarified gases, and at the intersection of a gas–fluid interface with a wall, it is violated. But at normal densities and pressures, and in homogeneous

fluids,* it is correct. Certainly, if one stands in a stiff breeze, the air appears to flow freely over one's face. Nevertheless, in the immediate vicinity of a stationary body, the air velocity tends to zero. It does so across a thin boundary layer – see Chapter 5. The sensation of airflow at the surface is due to pressure and heat transfer effects produced by the flow.

No-slip is certainly not an obvious condition. Early fluid dynamicists, including Stokes, were uncertain. The book *Hydrodynamics* by Horace Lamb was called the bible of fluid dynamics in the late 19th and early 20th centuries. It contains a section entitled on the "question of slipping." The matter was resolved initially by comparing the predictions of flow through a pipe, allowing for the possibility slip, to experiment. The data implied no-slip: $\boldsymbol{u} = 0$ at a stationary surface. Subsequently, high-resolution measurements of velocity proved no-slip beyond doubt.

The no-slip boundary condition is the origin of viscous drag: if the fluid away from the surface is in motion, and the velocity is brought to zero as the wall is approached, then there is a shearing stress imposed on the wall. This is illustrated by the simple examples of Couette and Poiseuille flow.

Interestingly, Poiseuille was a French physician – albeit with training in physics. His research on the flow of blood in arteries led him to investigate flow through narrow tubes. In 1838, he experimentally determined his famous law that the volume flow rate through a circular tube of radius a and length L is given by

$$Q = \frac{\Delta p}{8\mu L}\pi a^4, \tag{1.16}$$

where Δp is the pressure drop across the pipe. Stokes's mathematical derivation of this result was one piece of evidence in favor of the Navier–Stokes equations.

Couette flow is named for a French scientist who in 1890 measured the viscosity of liquids in what has come to be known as a Couette flow viscometer. Couette and Poiseuille flow are *parallel* shear flows in the sense that the flow is unidirectional. In Couette flow, the velocity is in the x direction and is a linear function of the cross-stream coordinate, y: the exact expression is simply

$$u(y) = U_w \frac{y}{H}. \tag{1.17}$$

This is the velocity between two flat walls, the lower being stationary and the upper moving with velocity U_w and being located at $y = H$. The shear is uniformly equal to $dU/dy = U_w/H$, and the viscous stress is μ times this. The no-slip condition requires fluid next to the upper wall to move at U_w and that next to the lower to remain stationary, hence producing the shear and the viscous shear stress. This is an exact, though trivially simple, solution of the Navier–Stokes Eq. (1.14).

The viscous stress is $\mu U_w/H$. Normalized by $1/2\rho U_w^2$, this becomes

$$\frac{2\mu U_w}{H\rho U_w^2} = \frac{2}{(U_w H/\nu)} = \frac{2}{Re}$$

* Even at interfaces, it is only in a tiny region around the three phase contact line that no-slip is violated.

Re, defined here to be $U_w H/\nu$, is the Reynolds number. It is nondimensional: the numerator has dimensions of velocity times length; the denominator has dimensions of length squared per time. The Reynolds number measures the ratio of viscous to inertial forces. In the context of Couette flow, it might not be appropriate to invoke the scaling $1/2\rho U^2$ for inertial forces, but that is the form suggested by Bernoulli's equation [Eq. (1.38)].

The other elementary example, Poiseuille flow, is *fully developed* flow in a pipe. Fully developed means that the velocity profile is unchanged with downstream distance. Hence, it is the flow in a straight pipe, well downstream of the inlet. When the flow is not fully developed, CFD can be used to compute the velocity profile, and to examine how it approaches a fully developed condition, if the pipe is long enough for that to occur.

The Poiseuille velocity, as a function of radius from the center of the pipe, is a parabola:

$$u = 2\overline{U}\left(1 - \frac{r^2}{a^2}\right), \tag{1.18}$$

where a is the radius and $\overline{U}$ is the cross-section-averaged velocity. The radial coordinate, r, is zero at the center and is a at the wall of the pipe. In §1.5, the parallel flow approximation will be described more fully. Parallel flow analysis shows that the cross-section-averaged velocity is related to the pressure gradient down the pipe by Poiseuille's law: $\overline{U} = -a^2(dP/dx)/8\mu$. This says that a net pressure drop, $dP/dx < 0$, is needed to overcome viscous friction and cause the fluid to flow. In a length L of pipe, this pressure drop is $8\mu\overline{U}L/a^2$.

Again, normalize the pressure drop by $1/2\rho U^2$ to compare viscous and inertial forces:

$$\frac{8\mu\overline{U}L}{1/2\rho\overline{U}^2a^2} = \frac{16(L/a)}{\overline{U}a/\nu} = \frac{16(L/a)}{Re}, \tag{1.19}$$

where $Re \equiv \overline{U}a/\nu$ defines the Reynolds number based on bulk velocity and pipe radius. The numerator is just a nondimensional statement that the pressure drop increases linearly with the length of the pipe, in units of pipe radii.

In both Couette and Poiseuille flows, the Reynolds number emerges as a measure of the role of viscous friction. The Reynolds number is named for the British engineer Osborne Reynolds. In an article published in 1883, he addressed the question of when flow through a pipe transitioned from a laminar to a turbulent state, concluding that it was characterized by the nondimensional parameter that was subsequently named for him. His analysis was largely concerned with resolving discrepancies between the Navier–Stokes equations and observations of the rate of volumetric discharge from pipes. He showed experimentally that the discrepancy was due to unsteadiness and estimated a critical value of his parameter at which the unsteadiness could be expected. His pioneering researches earned him credit for the most commonly cited, nondimensional parameter of fluid dynamics.

A brief, entertaining history of the term *Reynolds number* was written by Rott (1990). He credits Sommerfeld with coining the term in a 1908 article at the Fourth International Congress of Mathematicians in Rome. However, he credits Prandtl with popularizing the term through his writings in the 1910s. The more expansive view of Reynolds' parameter is that it embodies the notion of dynamic similarity. His initial motive of characterizing hydrodynamic instability is just one instance. Generally, the motion of two fluids of differing velocity will be similar if they are placed in geometrically similar apparatuses and flowed with a speed adjusted so that UL/ν is the same for both fluids. For instance, the velocity field of air or water that approaches cylinders of radii a_a and a_w at speeds U_a and U_w, respectively, will be identical if these speeds and diameters are such that $U_a a_a/\nu_a = U_w a_w/\nu_w$, provided the velocities are normalized by U_a and U_w.

This broader understanding of the significance of the Reynolds number is justified by putting the Navier–Stokes equation [Eq. (1.14)] into nondimensional form. If the nondimensional velocity is defined as $\tilde{u} = u/U_r$, where U_r is a reference velocity, and the nondimensional coordinate is defined by $\tilde{\boldsymbol{x}} = \boldsymbol{x}/a$, where a is a reference length, then it is found that the Navier–Stokes equation takes the form

$$Re\left[\frac{\partial \tilde{\boldsymbol{u}}}{\partial \tilde{t}} + (\tilde{\boldsymbol{u}} \cdot \nabla)\tilde{\boldsymbol{u}}\right] = -\nabla \tilde{p} + \nabla^2 \tilde{\boldsymbol{u}}. \tag{1.20}$$

The fluid properties enter only through the parameter *Re*. The solution, in any given geometry, is a function solely of *Re*. That is why the chapters of this book are organized into ranges of Reynolds number. The various behaviors of fluid flow are obtained by solving the governing equations with representative values of this parameter. This is referred to as dynamic similarity: if two flows have the same geometry and boundary conditions (in dimensionless form), and if they also have the same *Re*, their entire flow fields will be equivalent. For instance, it does not matter whether the fluid is water or air. The kinematic viscosity of air is 15 times that of water. The product of U and a must be 15 times larger in air to obtain dynamic similarity. In a computer simulation using a fixed grid, the fluid type and inlet velocity can be selected to achieve a desired *Re*.

Conversely, given the geometry, fluid type, and boundary conditions, the flow is a function of *Re* – often quite dramatically so. Flow phenomena as a function of Reynolds number will be discussed in Chapter 4.

Actually, Eq. (1.20) is the form obtained if pressure scales with viscosity as in the above solution for Poiseuille flow: $p \sim \mu U_r/a$. But if the pressure scales as ρU_r^2, then the Navier–Stokes equation assumes the form

$$\left[\frac{\partial \tilde{\boldsymbol{u}}}{\partial \tilde{t}} + (\tilde{\boldsymbol{u}} \cdot \nabla)\tilde{\boldsymbol{u}}\right] = -\nabla \tilde{p} + \frac{1}{Re}\nabla^2 \tilde{\boldsymbol{u}}. \tag{1.21}$$

Equation (1.20) gives an insight into flows with low Reynolds number, Eq. (1.21) gives an insight into high Reynolds number flow. In the former case, letting $Re \to 0$ leaves only the viscous and pressure forces. This is the limit of inertialess flow, which is the subject of Chapter 3.

In the case of Eq. (1.21), letting $Re \to \infty$ leaves a balance between pressure forces and convective acceleration. This is the limit discussed in Chapter 5, with one important caveat: viscous forces will always come into play very close to surfaces. The implications of that caveat will be explored in Chapter 5.

1.5 Parallel Flow Approximation

1.5.1 Couette–Poiseuille Flow

The Navier–Stokes equations have a few exact solutions. They are for very simple, idealized geometries. These exact solutions can provide estimates of quantities like pressure drop or frictional drag. They can provide a framework for less simple cases, which are computed numerically. One rather useful class of exact solutions is that of parallel flows.

Mathematically, a parallel flow is one in which the velocity does not vary in the direction of the flow. For instance, if the flow is in the x direction, the derivative of $\boldsymbol{u}$ in the direction of the flow is $(\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = u\partial \boldsymbol{u}/\partial x$. The operator $\boldsymbol{u} \cdot \nabla$ is the projection of the derivative into the direction of the flow. If $\boldsymbol{u}$ is only a function of y and z, this vanishes. The fully developed flow down a pipe is a velocity, u, as a function of radius, r, and hence is a parallel flow.

In steady flow, the particles move along streamlines. If the velocity does not change along the streamline, then the particle experiences no acceleration. The condition for the velocity to be constant along a streamline is that the distance to neighboring streamlines be constant: that is, that they be parallel. The reasoning is as follows: by definition, streamlines are in the direction of the fluid flow; there is no flow across them. Hence, the mass flow follows the streamlines. If the space between two streamlines were to decrease, the flow would have to accelerate to preserve the mass flux. Hence, zero acceleration requires parallel flow.

If the flow is steady and parallel, inertia vanishes, even if the Reynolds number is not small. Then the equations become linear: it is not surprising that parallel flow solutions play a large role in classical fluid mechanics.

When the acceleration vanishes, the Navier–Stokes momentum equations (1.11) become

$$\begin{aligned} 0 &= \nu\left(\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) - \frac{1}{\rho}\frac{\partial p}{\partial x}, \\ 0 &= \frac{\partial p}{\partial y}, \\ 0 &= \frac{\partial p}{\partial z}, \end{aligned} \tag{1.22}$$

for the case of flow in the x direction, as a function of y and z. If v and w are zero, the y and z derivatives of pressure must also be zero; hence, pressure can only be a function of x. But u is independent of x, so Eq. (1.22) implies that $\partial_x p$ cannot be a function of x and, hence, must be a constant.

Equation (1.22) describes the flow down a pipe of arbitrary cross section. When the pipe is circular, the velocity is a function of $r = \sqrt{x^2 + y^2}$. Equation (1.22) can be transformed into Eq. (E1.1) on p. 58. It will be left as an exercise for the reader to explore analyses of flow in circular cross-section pipes.

If u is a function only of y and $\partial_x p$ is constant, Eq. (1.22) is readily integrated to obtain

$$u = A + By + \frac{y^2}{2\mu}\partial_x p. \tag{1.23}$$

That is, the velocity is a second-order polynomial. This parabolic profile is sometimes used as the inlet condition for a computation. If the flow enters the domain through a channel, this might be a suitable prescription.

The integration constants in Eq. (1.23) are determined by boundary conditions. For instance, Couette flow (1.17), has zero pressure gradient and $u(0) = 0$, $u(H) = U_w$, giving $A = 0, B = U_w/H$. Plane Poiseuille flow has $u(0) = 0 = u(H)$ giving $A = 0$, $B = -H\partial_x p/2\mu$.

The availability of simple formulas for parallel flow makes it useful for approximate analysis. In the case of plane Poiseuille flow, the mass flux per unit width, $\dot{m}$, is

$$\dot{m} \equiv \int_0^H \rho u \, dy = \int_0^H \frac{y^2 - yH}{2\nu}\partial_x p = -\frac{H^3}{12\nu}\partial_x p \tag{1.24}$$

having substituted Eq. (1.23) with the appropriate values of A and B. This is the planar analogue to Poiseuille's law (1.18).

If flow through a duct is computed with prescribed inlet mass flow, the pressure drop can be estimated from this formula. For the purpose of estimation, let us allow H to be a function of x. Then, integrating gives

$$p_{\text{out}} - p_{\text{in}} = -12\nu \int_0^L \frac{\dot{m}}{H^3(x)} dx \tag{1.25}$$

for the pressure drop across the length from $x = 0$ to $x = L$.

Parallel flow as an approximation can be illustrated by computing the flow in a curved, contracting duct. The uppermost portion of Figure 1.2 shows the development of the velocity profile in the entrance to such a duct. Corresponding velocity vectors in the entrance region are show at the lower left. They start from the plug flow, $u =$ constant. The parabolic profile develops fairly close to the entrance in this case. The Reynolds number is the low value of 10. A standard estimate of the entrance length equates it to $0.06HRe$. In this case, the parabolic profile is established by $x \approx 0.6H$.

Pressure is assumed to depend only on x. That means that pressure contours should be vertical lines, as they are, except for a small departure in the curved region. The computed centerline pressure distribution is compared to the parallel flow approximation (1.25) at the lower right of Figure 1.2. The agreement is excellent, after the short entrance length. Consistent with Eq. (1.24), the pressure gradient

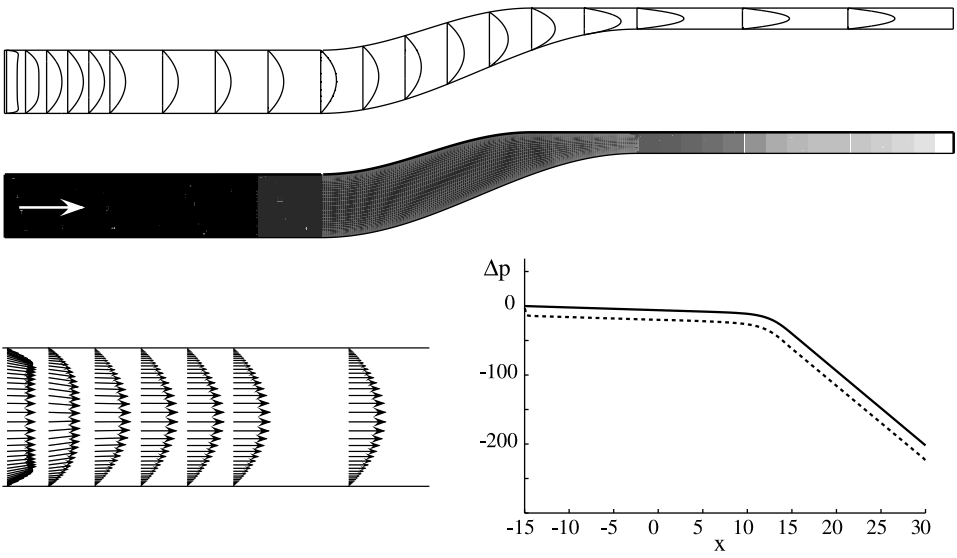


Figure 1.2. Quasiparallel flow in a duct. Pressure contours and velocity profiles in the entire domain; development of the velocity vectors in the entrance region. The computed centerline pressure distribution (– – – –) is compared to the theory (1.25) (———).

downstream of the bend is greater than the upstream pressure gradient by a factor of H^3; in this case H decreases by 1/3 and pressure gradient increases by 27.

1.5.1.1 Branching Tubes

Rather complex examples of flow in a system of tubes are found in the human body. The cardiovascular system consists of highly branched tubes, through which blood flows, with the heart, as its pump. The pulmonary system consists of highly branched tubes, through which air flows, pumped by the lungs. Let us apply Poiseuille's law to these systems; after all, it was blood flow in arteries that motivated his studies.

At some level of abstraction, human circulatory systems can be analyzed as networks of branching pipes. A single pipe may fork into two at a junction. On average the total cross-sectional area of the human arteries and, in the pulmonary system, the bronchia, increases as the individual tubes become smaller. Let A_1 be the area before a bifurcation. Let βA_1 be the area of each of two equal tubes leaving the bifurcation. Then the area ratio is 2β. If the total area increases, $\beta > 1/2$.

Poiseuille's law (1.16), for pressure drop in a circular tube, can be restated as

$$\Delta p = \frac{8\pi\mu QL}{A^2}, \tag{1.26}$$

where $A = \pi a^2$ is the cross-sectional area and $Q = \overline{U}\pi a^2$ is the volume flux. If the flow Q splits equally into two tubes, the volume flux in each is $1/2Q$, irrespective of their radii.

The pressure drop across a length L_1 of the larger tube and L_2 of the smaller tubes is

$$\frac{8\pi\mu Q}{A_1^2}\left[L_1 + \frac{L_2}{2\beta^2}\right],$$

wherein $\beta = A_2/A_1$. The pressure drop is a factor of

$$\frac{L_1 + L_2/2\beta^2}{L_1 + L_2}$$

larger than a single tube of the same total length. Bifurcation increases the pressure drop. Even if $\beta = 1/2$, so that the total area is unchanged, the pressure drop goes up. This is because the Reynolds number decreases. The Reynolds number can be expressed in terms of volume flux and area as

$$Re = \frac{Q}{\nu\sqrt{\pi A}}.$$

Hence $Re_2 = Re_1/(2\sqrt{\beta})$. Only if $\beta = 1/4$ does Re remain constant, but then the total area is not constant.

We have asserted that $\beta > 1/2$ in human circulatory systems. (That may be violated at the first level of bifurcation, but it is satisfied on average.) After n bifurcations, the Reynolds number has fallen by $(4\beta)^{n/2}$. In the larger arteries and airways of the lung, Reynolds numbers may be in the range of a few hundred to a few thousand (Lighthill, 1975). Although the higher values may verge on transition to turbulence, after one or two levels of bifurcation, only laminar flow can be expected; human arteries branch approximately 20 times. It can also be expected that the smaller arteries will dominate the pressure drop. If tube length scales in the same way as tube radius, that is, $L_2 = \sqrt{\beta}L_1$, then the smaller tubes contribute $1/(2\beta^{3/2})$ times the pressure drop of the larger tube. Generally, this will be greater than unity, but if $\beta = 2^{-2/3}$ the pressure drop will be the same across each generation of the system.

The assumption of Poiseuille flow is not likely to be good near to the bifurcation. It will take some distance for the parabolic profile to be established. In straight pipes, this entrance length is estimated as $0.24aRe_a$ – that is, the condition for 5% accuracy of the centerline velocity, starting from plug flow. In the entrance region, the profile is steeper, and the wall stress is higher, than in fully developed flow (Figure 1.2). At a bifurcation, the stress is especially large at the junction. In the larger arteries and bronchia, where the Reynolds number can be high, the entire length of tube may be in the entrance region. Resistance will be higher than predicted from Poiseuille flow estimates. After a few levels of branching, the entrance length will be less than the length of the tube.

1.5.1.2 Circular Couette Flow

Idealized, parallel flows are commonly used for approximate analysis. To illustrate, consider flow in the annulus between two concentric circular cylinders, as in Figure 1.3 at the left. At the right, the geometry is modified to make it be a very small pump that

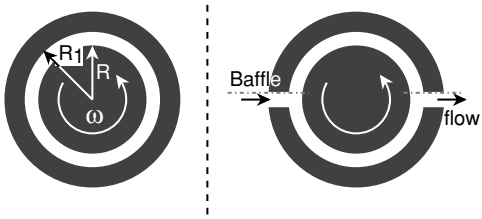


Figure 1.3. Circular Couette flow as a model.

operates by simply dragging fluid with a rotating shaft. A baffle prevents backflow. The geometry at left provides the parallel flow solution; it is used to model the situation at right.

The configuration at left is called circular Couette flow. From it, estimates of the volume flow rate (per unit length), Q, and of the torque exerted by the rotating shaft will be obtained.

The concentric circles have radii R and R_1, and the inner cylinder rotates with angular velocity ω. The flow is parallel because the velocity is in the angular direction and is only a function of radius. Let that velocity be $u_\theta(r)$. There can be no pressure gradient in the angular direction because of the circular symmetry. (If there were a gradient, it would have to be constant; but then the pressure at $\theta = 0$ would not equal that at $\theta = 2\pi$.) Because of the circular geometry, Eq. (1.22) takes the form

$$0 = \nu \frac{\partial}{\partial r}\left[\frac{1}{r}\frac{\partial (r u_\theta)}{\partial r}\right],$$

$$\frac{u_\theta^2}{r} = \frac{\partial p}{\partial r}.$$

The first is the form that the viscous force takes when expressed in cylindrical coordinates and in terms of the angular velocity. The general solution is just

$$u_\theta = \frac{A}{r} + Br$$

instead of the parabolic profile, Eq. (1.23), that obtains in planar flow. The conditions, $u_\theta = \omega R$ and $u_\theta = 0$ on the inner and outer cylinders, are met by

$$u_\theta = \frac{\omega R^2}{R^2 - R_1^2}\left(r - \frac{R_1^2}{r}\right).$$

The torque on the inner cylinder follows from Eq. (1.7). From that formula, the force in the angular direction is $2\pi R \sigma_{r\theta}$. The area of the wall is $2\pi R$ times width, which gives the first factor. The second is the viscous shear stress on the wall. The torque is R times this. In cylindrical coordinates, the formula for viscous stress is (White, 1991)

$$\sigma_{r\theta} = \mu r \frac{\partial}{\partial r}\left(\frac{u_\theta}{r}\right)$$

giving

$$4\pi\mu\omega R^2 R_1^2/(R^2 - R_1^2)$$

for the torque (per unit width) exerted *by* the cylinder *on* the fluid.

The volume flux (per unit width) that is dragged around in between the cylinders is

$$Q \equiv \int_R^{R_1} u_\theta dr = \frac{\omega R^2}{R^2 - R_1^2}\left[1/2(R^2 - R_1^2) - R_1^2 \log(R/R_1)\right].$$

For fixed R_1, the maximum of this formula for volume flow occurs when $R = 0.562\,R_1$, for which $Q = 0.108\omega R_1^2$. This is the maximum rate at which fluid can be pumped for a fixed shaft speed and outer radius.

Power is torque times angular velocity. With the above expression for torque, it is found that the power is

$$\text{Power} = 4\pi\mu\omega^2 \frac{R^2 R_1^2}{(R_1^2 - R^2)} = 4\pi\mu\omega^2 R_1^2 \times 0.4621,$$

when $R = 0.562 R_1$.

1.5.2 Oscillatory Boundary Layer

Another example of parallel flow is Stokes's oscillatory boundary layer. Consider a flat plate that is oscillated horizontally beneath a quiescent fluid. The plate is the plane $y = 0$, and it is oscillated in the x direction. First consider what is expected to happen. As the plate moves from left to right, it will drag the fluid immediately above it. That is because of the no-slip boundary condition. Viscosity will diffuse that movement higher into the fluid. A layer of fluid will be moving from left to right with the plate. The plate reaches its maximum displacement to the right and starts to return to the left. Again, fluid next to the wall is dragged along. Now there is a layer moving to the left with the plate. Above that, the fluid is still moving to the right, because leftward momentum has not yet diffused to that height. Following this reasoning for a few periods of oscillation leads to the conclusion that layers of alternatively left- and right-moving fluid will be created above the oscillating plate. That is indeed the case. These layers diffuse into one another, so as the height above the plate increases, the magnitude of the directional oscillations decreases.

The mathematical analysis consists in solving

$$\frac{\partial u}{\partial t} = \nu \frac{\partial^2 u}{\partial y^2}. \tag{1.27}$$

This is the unsteady parallel flow equation. At $y = 0$, the boundary condition is $u = U_w \sin(\omega t)$. As $y \to \infty$, $u \to 0$. The solution is

$$u = U_w e^{-y\sqrt{\omega/2\nu}} \sin(\omega t - y\sqrt{\omega/2\nu}),$$

or, if the layer thickness $\delta = \sqrt{2\nu/\omega}$ is introduced as an abbreviation,

$$u = U_w e^{-y/\delta} \sin(\omega t - y/\delta). \tag{1.28}$$

This embodies the physical picture of the previous paragraph. At any given time, the direction of the flow oscillates with height. The amplitude of the oscillations

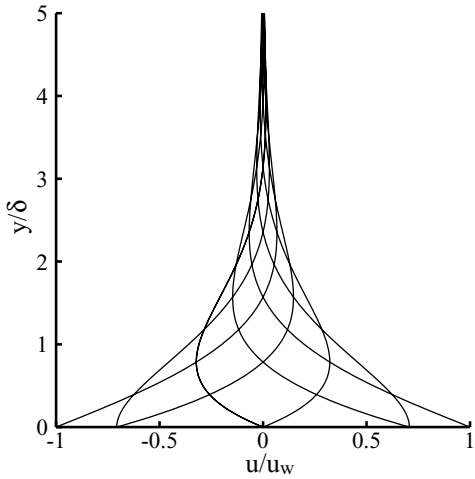


Figure 1.4. Velocity profiles at various times in Stokes's oscillatory boundary layer.

decays exponentially with height, as is seen in Figure 1.4. The exponential damping length scale is $\sqrt{2\nu/\omega}$. Lower frequencies penetrate higher above the wall than higher frequencies.

Viscous force is exerted by a moving wall. Work is done, at a rate equal to force times velocity. The viscous force, per unit area, is

$$\mu \left.\frac{\partial u}{\partial y}\right|_{y=0} = -\frac{\mu U_w}{\delta}[\cos(\omega t) + \sin(\omega t)].$$

The rate of work is this times the wall velocity:

$$\begin{aligned} F \cdot U_w &= -\frac{\mu U_w^2}{\delta}\left[\sin(\omega t)\cos(\omega t) + \sin^2(\omega t)\right] \\ &= -\frac{\mu U_w^2}{2\delta}\left[\sin(2\omega t) + 1 - \cos(2\omega t)\right]. \end{aligned}$$

The sinusoidal terms average over time to zero. Denoting the averaged rate of work by an overbar,

$$\overline{F \cdot U}_w = -\frac{\mu U_w^2}{2\delta}. \tag{1.29}$$

This, times the wall area, is the mean rate at which the wall does work on the fluid.

Where does that energy go? It is dissipated by friction. To see this, multiply Eq. (1.27) by ρu to obtain the energy equation

$$1/2\rho\frac{\partial u^2}{\partial t} = -\mu\frac{\partial u}{\partial y}\frac{\partial u}{\partial y} + \mu\frac{\partial}{\partial y}\left(u\frac{\partial u}{\partial y}\right).$$

An average of the left side over a period of oscillation vanishes because it is a time derivative: integrating is over one period gives $1/2\rho\left[u^2(t_1) - u^2(t_2)\right]$, where t_1 and t_2 are separated by one period, so that the velocities are equal and their difference is zero. The two terms on the left are the dissipation and diffusion of energy. The first is called dissipation because it is negative definite: it is viscosity times the square of

the velocity gradient preceded by a minus sign. It can be only negative or zero. It will always be a sink of energy.

Integrating the two terms on the right from the wall to infinity in y and equating their time-average to zero gives

$$\overline{\mu \frac{\partial u}{\partial y} u}\Bigg|_0 = \overline{F \cdot U}_w = \int_0^\infty \mu \overline{\left(\frac{\partial u}{\partial y}\right)^2} dy. \tag{1.30}$$

The leftmost term is recognized as the viscous force per unit of wall area times the wall velocity. The rightmost term is the total rate of energy dissipation within the fluid. The mathematics supports an understanding that work done on the fluid by the oscillating wall is balanced by dissipation interior to the flow. The dissipated energy would show up as heat.

An oscillatory flow also could be driven by an oscillating pressure gradient with a stationary wall (exercise 1.9). The solution is again Eq. (1.28) with the velocity $U_w \sin(\omega t)$ subtracted. U_w is now the velocity in the freestream. Energy dissipation depends only on the velocity gradient, so it remains unchanged; Eq. (1.29) is still valid. This is the rate at which energy must be supplied to maintain the oscillations.

An example of driven oscillations in a pipe is proposed in exercise 1.6. For Eq. (1.29) to be used in oscillating pipe flow, δ would have to be small compared to the pipe diameter. That is because it is a solution for a plane wall in an unbounded fluid. δ is small when the frequency is high. If the pipe diameter is d, the condition $\delta \ll d$ is equivalent to $\omega d^2/\nu \gg 1$. If that condition is met, multiplying Eq. (1.29) by the total area of the wall provides an estimate of the rate of energy input that is needed to oscillate the flow against the retarding effect of viscous friction.

In §8.3.2 it will be explained how Eq. (1.29) provides an estimate of the rate at which free oscillations of a liquid sloshing in a container are damped. Dissipation is larger than might naively be supposed. It occurs in a layer of thickness δ. The naive supposition would be that in a container of radius a dissipation is spread through out the container, and hence $\overline{F \cdot U}_w = -\mu U_w^2/2a$. That is suitable at low frequencies, but Eq. (1.29) is correct when the frequency is high. The naive estimate is a factor of $\delta/a = \sqrt{2\nu/\omega a^2}$ smaller than the actual rate. At high frequency, the rate of energy dissipation can be very much greater than at first anticipated.

The parallel flow assumption is fairly restrictive. Without it, closed form solutions of the viscous equations are rarely possible. However, general considerations can be advanced that guide the study of fluid flow. They are the topic of §1.6.

1.6 Some Basics

Introductory fluid dynamics starts from the global conservation laws applied to a finite-sized control volume. Eventually, it leads to the differential form of the laws that apply to infinitesimal control volumes. As the differential laws have already been described, we proceed in the other direction to derive the global conservation equations from the differential form. The global conservation equations do not do

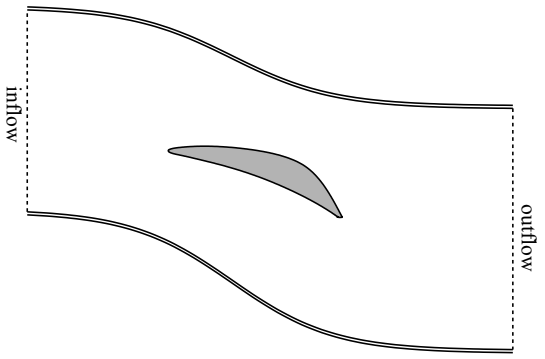


Figure 1.5. Schematic for one-dimensional conservation equations.

justice to the complexity of fluid velocity fields. However, they serve to fix ideas about the nature of fluid forces, about advection of momentum, and about the relation between fluid dynamics and forces on obstacles in fluid flow.

1.6.1 One-Dimensional Conservation Laws

The one-dimensional conservation laws, studied in introductory fluid dynamics, can be seen as the integral of the momentum and mass conservation laws (1.11) and (1.15) over a large control volume. This elementary statement of the governing laws is useful for designing computations, for making engineering estimates, and as a check on computations. They are summarized here as a reminder and for future reference; it is expected that the reader already has some familiarity with these basics.

Consider a flow domain consisting of inflow and outflow boundaries and surfaces – schematically, something like Figure 1.5. The divergence theorem of vector calculus can be used to integrate Eqs. (1.11) and (1.15) over the domain. The formal statement of this theorem is

$$\int_{\mathcal{V}} \nabla \cdot \boldsymbol{F} d\mathcal{V} = \int_{\mathcal{S}} \boldsymbol{F} \cdot \hat{n} d\mathcal{S}. \tag{1.31}$$

The integrals are over the volume and surface of any region. $\hat{n}$ is the outward normal to the surface. This basically is an elaboration of the fundamental theorem of calculus that

$$\int_{x_1}^{x_2} \frac{df}{dx} dx = f(x_2) - f(x_1).$$

The outward normal to the interval (x_1, x_2) is in the $+x$ direction at x_2 and in the $-x$ direction at x_1. Hence the right side of this equation is analogous to the right side of Eq. (1.31). The theorem applies to any volume of integration. In the case of Figure 1.5, it is the entire macroscopic region: applying the divergence theorem gives rise to the integrated, control volume equation that will be discussed shortly. In the case of a computer algorithm, the volume is a small computational cell, as will be described in Chapter 2.

First consider the continuity Eq. (1.15). Integrating over a control volume and applying the divergence theorem produces the equation of net mass conservation

$$0 = \int_{\mathcal{V}} \nabla \cdot \boldsymbol{u} d\mathcal{V} = \int_{\mathcal{S}} \boldsymbol{u} \cdot d\boldsymbol{A},$$

where the notation $d\boldsymbol{A} = \hat{n} dS$ is introduced. Hence, in the case of Figure 1.5,

$$-\int_{\text{inflow}} \rho \boldsymbol{u} \cdot d\boldsymbol{A} = \int_{\text{outflow}} \rho \boldsymbol{u} \cdot d\boldsymbol{A}. \tag{1.32}$$

$\boldsymbol{A}$ in this equation is an area vector on the boundary, pointing outward from the flow domain. Thus Eq. (1.32) states that the mass flow out of the domain is equal to the mass flow into the domain. The minus sign arises on the left because $\boldsymbol{A}$ points out of the domain.

If u can be treated as constant on the inflow and outflow boundaries, Eq. (1.32) simplifies to the well-known, one-dimensional conservation equation

$$\rho u A|_{\text{in}} = \rho u A|_{\text{out}} \equiv \dot{m}. \tag{1.33}$$

In Eq. (1.33), u is the perpendicular component of the velocity crossing the boundary and A is the area of the boundary. $\dot{m}$ is the mass flux, which is the same across the inflow or outflow surfaces. For instance, if the outflow area is twice the inflow, and density is constant, then the velocity will fall by a factor of 2 from inlet to exit. A more correct statement is that the velocity averaged over the cross section falls by a factor of 2. The assumption that u is uniform will usually be overly crude. Nevertheless, it becomes needed when the momentum equation is analyzed.

The continuity equation allows the momentum equation (1.11) to be written in conservation form as

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

If the flow is steady in time, an integral of this equation gives

$$\int_{\text{inflow}} (p d\boldsymbol{A} + \rho \boldsymbol{u}\boldsymbol{u} \cdot d\boldsymbol{A}) + \int_{\text{outflow}} (p d\boldsymbol{A} + \rho \boldsymbol{u}\boldsymbol{u} \cdot d\boldsymbol{A}) = \int_{\text{surfaces}} \sigma \cdot d\boldsymbol{A} \tag{1.34}$$

having, again, made use of the divergence theorem. This is a statement that the sum of the forces on all surfaces internal to the domain must be balanced by the a combination of net momentum flow into the domain and pressure difference across the domain. It is assumed that viscous stresses on the inflow and outflow boundaries are negligible – so that σ contributes $-p$. The balance is stated here for time-independent flow. If the the flow is unsteady, the rate of change of momentum inside the control volume must be added in the left side of Eq. (1.34).

On the right side of Eq. (1.34), the area vector points into the surface: we have integrated over a fluid control volume; the direction outward to that volume points into the surface. In Eq. (1.7), the area vector points out of the surface because force on the surface is being computed. Hence, the right side of Eq. (1.34) is minus the surface force.