1. Introduction

The development of modern fluid mechanics is closely connected to the name of its founder, Ludwig Prandtl. In 1904 it was his famous article on fluid motion with very small friction that introduced boundary-layer theory. His article on airfoil theory, published the following decade, formed the basis for the calculation of friction drag, heat transfer, and flow separation. He introduced fundamental ideas on the modeling of turbulent flows with the Prandtl mixing length for turbulent momentum exchange. His work on gas dynamics, such as the Prandtl–Glauert correction for compressible flows, the theory of shock waves and expansion waves, as well as the first photographs of supersonic flows in nozzles, reshaped this research area. He applied the methods of fluid mechanics to meteorology, and was also pioneering in his contributions to problems of elasticity, plasticity, and rheology.

Prandtl was particularly successful in bringing together theory and experiment, with the experiments serving to verify his theoretical ideas. It was this that gave Prandtl’s experiments their importance and precision. His famous experiment with the tripwire, through which he discovered the turbulent boundary layer and the effect of turbulence on flow separation, is one example. The tripwire was not merely inspiration, but rather was the result of consideration of discrepancies in Eiffel’s drag measurements on spheres. Two experiments with different tripwire positions were enough to establish the generation of turbulence and its effect on the flow separation. For his experiments Prandtl developed wind tunnels and measuring apparatus, such as the Göttingen wind tunnel and the Prandtl stagnation tube. His scientific results often seem to be intuitive, with the mathematical derivation present only to serve the physical understanding, although it then does indeed deliver the decisive result and the simplified physical model. According to Werner Heisenberg, Prandtl was able to “see” the solutions of differential equations without calculating them.

Selected individual examples aim to introduce the reader to the path to understanding of fluid mechanics prepared by Prandtl and to the contents and modeling in each chapter. As an example of the dynamics of flows (Chapter 4), the different regimes in the flow past a vehicle, an incompressible flow, and in the flow past an automobile, a compressible flow, are described.
In flow past a vehicle, we differentiate between the free flow over the surface and the flow between the vehicle moving with velocity $U_\infty$ and the street which is at rest. At the stagnation point, where the pressure is at its maximum, the flow divides, and is accelerated along the hood and past the spoiler along the base of the vehicle. This leads to a pressure drop and to a negative downward pressure on the street, as shown in Figure 1.1. The flow again slows down at the windshield, and is decelerated downstream along the roof and the trunk. This leads to a pressure increase with a positive lift, while the negative downward pressure on the street along the lower side of the vehicle remains.

Viscous flow (Section 4.2) on the upper and lower sides of the vehicle is restricted to the boundary-layer flow, which becomes the viscous wake at the back edge of the vehicle. In the wind tunnel experiment the flow is made visible with smoke, and this shows that downstream from the back of the automobile, a backflow region forms. This is seen in the figure as the black region. Outside the boundary layer and the wake, the flow is essentially inviscid (Section 4.1).

In order to be able to understand the different flow regimes, and therefore to establish a basis for the aerodynamic design of a motor vehicle, Prandtl worked out the carefully prepared path (Chapters 2 to 4) from the properties of liquids and gases, to kinematics, and to the dynamics of inviscid and viscous flows. By following this path, too, the reader will successively gain physical understanding of this first flow example.

The second flow example considers compressible flow past a wing with a shock wave (Sections 4.3 and 4.4.5). The free flow toward the wing has the

Fig. 1.1. Flow past a vehicle
velocity of a civil aircraft \( U_\infty \), a large subsonic velocity. Figure 1.2 shows the flow regimes on a cross-section of the wing and the negative pressure distribution, with the flow again made visible with small particles. From the stagnation point, the stagnation line bifurcates to follow the suction side (upper side) and the pressure side (lower side) of the wing. On the upper side, the flow is accelerated up to supersonic velocities, an effect that is connected with a large pressure drop. Further downstream, the flow is again decelerated to the subsonic regime via a compression shock wave. This shock wave interacts with the boundary layer and causes it to thicken, leading to increased drag.

On the lower side the flow is also accelerated from the stagnation point. However, in the nose region the acceleration is not as great as on the suction side, and so no supersonic velocities occur along the pressure side. From about the middle of the wing onwards, the flow is again decelerated. The pressures above and below then approach one another, leading to the wake region downstream of the trailing edge.

A thin boundary layer is formed on the suction and pressure sides of the wing. The suction and pressure side boundary layers meet at the trailing edge and form the wake flow downstream. As in the example of the flow past a motor vehicle, both the flow in the boundary layers and the flow in the wake are viscous. Outside these regions the flow is essentially inviscid.

The pressure distribution in Figure 1.2 results in a lift, which, for the wing of the civil aircraft, has to be adapted to the number of passengers to be transported. In designing the wing, the design engineer has to keep the

---

**Fig. 1.2.** Flow past a wing
drag of the wing as small as possible to save fuel. This is done by shaping the wing appropriately.

Different equations for computing each flow result from the different properties of each flow regime. To good approximation, the boundary-layer equations hold in the boundary-layer regime. In contrast, computing the wake flow and the flow close to the trailing edge is more difficult. In these regimes, the Navier–Stokes equations have to be solved. The inviscid flow in the region in front of the shock can be treated using the potential equation, a comparatively simple task. The inviscid flow behind the shock outside the boundary layer has to be computed with the Euler equations, since the flow there is rotational. In the shock boundary-layer interaction region, again the Navier–Stokes equations have to be solved.

In contrast to Prandtl’s day, numerical software is now available for solving the different partial differential equations. Because of this, in Chapter 5 we present the fundamental equations of laminar and turbulent flows as a basis for the following chapters dealing with the different branches of fluid mechanics. Following the same procedure as Prandtl, the mathematical solution algorithms and methods may be found by referral to the texts and literature cited.

As will be shown in Chapters 6 to 12, notwithstanding of numerically computed flow fields, it is necessary to consider the physical modeling in the different regimes. There are still no closed theories of turbulent flows, of multiphase flows, or of the coupling of flows with chemical reactions out of thermal or chemical equilibrium. For this reason, Prandtl’s method of intuitive connection of theory and experiment to physical modeling is still very much up-to-date.

The fascinating complexity of turbulence has attracted the attention of scientists for centuries (Chapter 6). For example, the swirling motion of fluids that occurs irregularly in space and time is called turbulence. However, this randomness, is not without some order, as is apparent from casual observation. Turbulent flows are a paradigm for spatially extended nonlinear dissipative systems in which many length scales are excited simultaneously and coupled strongly. The phenomenon has been studied extensively in engineering and in such diverse fields as astrophysics, oceanography, and meteorology.

Figure 1.3 shows a turbulent jet of water emerging from a circular orifice into a tank of still water. The fluid from the orifice is made visible by mixing small amounts of a fluorescing dye and illuminating it with a thin light sheet. The picture illustrates swirling structures of various sizes amidst an avalanche of complexity. The boundary between the turbulent flow and the ambient is usually rather sharp and convoluted on many scales. The object of study is often an ensemble average of many such realizations. Such averages obliterate most of the interesting aspects seen here, and produce a smooth object that grows linearly with distance downstream. Even in such smooth objects, the averages vary along the length and width of the flow, these variations being a
measure of the spatial inhomogeneity of the turbulence. The inhomogeneity is typically stronger along the smaller dimension of the flow. The fluid velocity measured at any point in the flow is an irregular function of time. The degree of order is not as apparent in time traces as in spatial cuts, and a range of intermediate scales behaves like fractional Brownian motion.

In contrast, Figure 1.4 shows homogeneous and isotropic turbulence produced by sweeping a grid of bars at a uniform speed through a tank of still water. Unlike the jet turbulence of Figure 1.3, turbulence here does not have a preferred direction or orientation. On average, it does not possess significant spatial inhomogeneities or anisotropies. The strength of the structures, such as they are, is weak in comparison with such structures in Figure 1.3. Homogeneous and isotropic turbulence offers considerable theoretical simplifications, and is the object of many studies.

In many fluid-mechanical problems, the onset of turbulent flow is due to instabilities. An example of this is thermal cellular convection in a horizontal fluid layer heated from below and under the effect of gravity. The base beneath the fluid has a higher temperature than the free surface. If a critical temperature difference between the free surface and the base is exceeded, the fluid is suddenly set into motion and, as in Figure 1.5, it forms hexagonal cell structures in the center of which fluid rises and on whose edges the fluid sinks. The phenomenon is known as thermal cellular convection. If the fluid

Fig. 1.3. Turbulent jet of water

Fig. 1.4. Homogeneous and isotropic turbulent flow
is covered by a plate, instead of hexagonal cells periodically spaced rolling structures are formed without surface tension. The reason for the instabilities is the same in both cases. Cold, denser fluid is layered above warmer fluid, and this tends to flow toward lower layers. The smallest perturbation to this layering leads to the onset of the equalizing motion, providing critical temperature difference is exceeded.

The transition to turbulent convection flow takes place with increasing temperature difference via several time-dependent intermediate states. The size of the hexagonal structures or the long convection rolls changes, but the original cellular structure of the instability can still be seen in the turbulent convection flow.

**Convection flows with heat and mass transport** are treated in Chapter 7. These occur frequently in nature and technology, and it is via such flows that heat exchange in the atmosphere determines the weather. The example of a tropical cyclone is shown in Figure 1.10. The extensive heat adjustment between the equator and the North Pole leads to convection flows in the oceans, such as the Gulf Stream (Figure 1.11). Convection flows in the center of the Earth are also the cause of continental drift and are responsible for the Earth’s magnetic field. In energy technology and environmental technology flows are connected with heat and mass transport, and with phase transitions, as in steam generators and condensers. Convection flows are used in cooling

*Fig. 1.5. Thermal cellular convection*
towers to transport the waste heat from power stations. Other examples of convection flows are the propagation of waste air and gas in the atmosphere and of cooling and waste water in lakes, rivers, and oceans, heating systems and air-conditioning in buildings, circulation of fluids in solar collectors and heat accumulators.

Figure 1.6 shows experimental results on thermal convection flows. In contrast to forced convection flows, these are free convection flows, where the flow is due to only lift forces. These may be caused by temperature or concentration gradients in the gravitational field. A heated horizontal circular cylinder initially generates a rising laminar convection flow in the surrounding medium, which is at rest, until the transition to turbulent convection flow is caused by thermal instabilities. Similar thermal convection flows occur at vertical and horizontal heated plates.

Multiphase flow (Chapter 8) is the flow form that appears most frequently in nature and technology. Here the word phase is meant in the thermodynamic sense and implies either the solid, liquid, or gaseous state, any of which can occur simultaneously in a one-component or multicomponent system of substances. Impressive examples of multiphase flows in nature are storm clouds containing raindrops and hailstones, and snow dust in an avalanche or a cloud of volcano ash.

In power station engineering and chemical process engineering, multiphase flows are an important means of transporting heat and material. Two-phase, or binary, flows determine the processes in the steam generators, condensers, and cooling towers of steam power stations. The rain from the cooling water of a wet cooling tower is shown in Figure 1.7. The water drops lose their heat by evaporation to the warmed rising air. Multiphase, multicomponent flows are used in the extraction, transportation, and processing of oil and natural gas. Such flow forms are also very much involved in distillation and rectification.
processes in the chemical industry. They also appear as cavitation effects on underwater wing surfaces in fast flows. The example in Figure 1.8 shows a cavitating underwater foil. Phenomena of this kind are highly undesirable in flow machinery since they can lead to serious material damage.

_Turbulent reactive flows_ (Chapter 9) are very important for a great number of applications in energy, chemical, and combustion technology. The optimization of these processes places great demands on the accuracy of the numerical simulation of turbulent flows. Because of the complexity of the interaction between turbulent flow, molecular diffusion, and chemical reaction kinetics, there is a great need for improved models to describe these processes.

Turbulent flames are characterized by a wide spectrum of time and length scales. The typical length scales of the turbulence extend from the dimensions of the combustion chamber right down to the smallest vortex in which turbulent kinetic energy is dissipated. The chemical reactions that cause the combustion have a wide spectrum of time scales. Depending on the overlapping of the turbulent time scales with the chemical time scales, there are regimes with a strong or weak interaction between chemistry and turbulence. Because of this, a joint description of turbulent diffusion flames generally always requires an understanding of turbulent mixing and combustion.
A complete description of turbulent flames therefore has to resolve all scales from the smallest to the largest, which is why a numerical simulation of technical combustion systems is not possible on today’s computers and why averaging techniques in the form of turbulence models have to be used. However, if turbulence models are to describe such aspects of technical application as mixing, combustion, and formation of emissions realistically, it is necessary to be able to better determine the parameters of such models from detailed investigations.

One promising approach is the use of direct numerical simulation, the generation of artificial laminar and turbulent flames with the computer. For a small spatial area, the conservation equations for reactive flows are solved, taking all turbulent fluctuations into account, and thus describing a small but realistic section of a flame. This can then be used to describe real flames. The formation of closed regions of fresh gas that penetrate into the exhaust are an interesting phenomenon of turbulent premixed flames. The time resolution of this transient process can be investigated by means of direct numerical simulation and is important in determining the region of validity of current models and in the development of new models to describe turbulent combustion. Figure 1.9 shows the concentration of OH and CO radicals, as well as the vortex strength in a turbulent methane premixed flame.

Many different flows in nature (Chapter 10) can be seen on Earth and in space. The flow processes in the atmosphere range from small winds to the tropospherical jet stream of strong winds surrounding the globe. One particularly impressive atmospheric phenomenon is the tropical cyclone, known in the Caribbean and the United States under the name hurricane. Hurricanes form in the summer months above the warm waters off the African coast close to the equator and move with a southeasterly flow first toward the Caribbean and then northeasterwards along the east coast of the United States. Wind speeds of up to 300 km/h can occur in these tropical wind storms, with much resulting damage on land. An example of a cyclone is shown in Figure 1.10. This figure shows the path and a satellite image of Hurricanes Ivan and

![Fig. 1.9. Turbulent premixed methane flame](image-url)
Charley which passed over the Caribbean islands and the southeast coast of the United States in 2004, and continued their path as a low-pressure region across the Atlantic as far as Europe.

The flow processes in the ocean extend from small phenomena such as water waves to large sea currents. An example of the latter is the Gulf Stream, which as a warm surface current can be tracked practically from the African coast, past the Caribbean to western and northern Europe. Thanks to its relatively high water temperature, it ensures a mild climate along the coast of Britain and Norway. In order to compensate the warm surface current directed towards the pole, a cold deep current forms, and this flows from the north Atlantic along the east coast of North and South America, toward the south. Both of these large flow systems are shown in Figure 1.11.

Microflows, a new area of fluid mechanics, are discussed in Chapter 11. Through advances in manufacturing technology, the flow processes and transport processes through microchannels and past micro-objects are becoming relevant for technical applications. Modern manufacturing methods permit very small structures considerably less than one millimeter in size to be made.
from various materials such as silicon, glass, metal or plastic. Complex fluidic functions then take place in tiny spaces.

An inkjet printer head is an example of a microfluidic system. The ink is ejected through a matrix of apertures about 45 \( \mu m \) diameter and generates points of color on the paper. Figure 1.12 shows the ejection of a single droplet of ink from the printer head. The pressure is built up in the cavity by piezo crystals or through application of heat and evaporation. Similar systems are used for highly precise dosage in process engineering.

In a second example, the favorable surface to volume ratio in microchannels is used to construct a compact micro heat exchanger. Figure 1.12 shows a crossflow heat exchanger made of a pile of metal sheets with microchannels of cross-section 100 x 200\( \mu m \) etched onto it. In a cube of side 14mm at temperature differences of up to 80\( K \), heat can be transferred at rates of up to 14\( kW \). The large transfer surface is advantageous not only for heat transfer, but can also be used in catalytic coating to improve material transfer in chemical reactions. Similar heat exchangers can be used as microreactors, where the temperature of the chemical reaction in a passage can be controlled very precisely by a heat carrier in a second passage. Chemical reactions that otherwise would be quite impossible can thus be made possible or optimized.

Depending on the fluid, flows through and past very small geometries cannot be treated using continuum mechanics. Corrections to the continuum mechanical equations or even molecular methods are necessary to represent correctly the physics of flows at these small length scales.

![Fig. 1.12. Examples of microfluidic components](image)
In contrast to the previous examples of flows, biofluid mechanics in Chapter 12 deals with flows that are characterized by flexible biological surfaces. One distinguishes between flows past animals in the air or in water, such as a bird in flight or a fish swimming, and internal flows, such as the closed human blood circulation.

The human heart consists of two separate pump chambers, the left and right ventricles. The right ventricle is filled with blood low in oxygen from the circulation around the body, and on contraction it is emptied into the lung circulatory system. The reoxygenated blood in the lung is passed into the circulation around the body via the left ventricle. A simple representation of the flow throughout one cardiac cycle is shown in Figure 1.13. The atria and ventricles of the heart are separated by the atrioventricular valves, which regulate the flow into the ventricles. They prevent backward flow of the blood during contraction of the ventricles. During relaxation of the ventricles, the pulmonary valves prevent backward flow of the blood out of the lung arteries, while the aortic valves prevent backward flow out of the aorta into the left ventricle.

During the cardiac cycles, the ventricles undergo periodic contraction and relaxation, ensuring the pulsing blood flow in the circulatory system around the body. This pump cycle is associated with changes in pressure in the ven-

![Flow simulation of the left heart ventricle, atrium and aorta](image)

Fig. 1.13. Flow of the human heart
tricles and arteries. The pressure differences control the opening and closing of the cardiac valves. In a healthy heart, the pulsing flow is laminar and does not separate. Defects in the pumping behavior of the heart and heart failure lead to turbulent flow regimes and backflow in the ventricles, increasing flow losses in the heart.

The flow simulation of Figure 1.13 shows the streamlines of the inward flow in the left ventricle accompanied by a ring vortex. The mitral valve is open and the aortic valve closed. Large inward flow velocities directed downward and with a maximal velocity of about 0.5 m/s can be seen. After a quarter of the cardiac cycle the ring vortex branches, and the blood begins to flow through the top of the ventricle. When the ventricle contracts, the aortic and mitral valves are closed. The left ventricle is completely filled with blood, and the flow velocities calculated are very small. As the blood flows out of the ventricle, the mitral valve is closed and the aortic valve open. The streamlines show the blood flow jet into the aorta. As the ventricle relaxes, both cardiac valves are closed. The flow into the left atrium can be seen.
Prandtl-Essentials of Fluid Mechanics
Oertel jr., H. (Ed.)
2010, XII, 795 p., Hardcover
ISBN: 978-1-4419-1563-4