Delft University of Technology

Lecture notes

CTB2110

Fluid Mechanics

C. Chassagne & T.S. van den Bremer

2022-2023 Q1

Contents

0 Introduction

0.1 Fluid Mechanics for Civil Engineers

The study of fluids in Civil Engineering is primarily linked to the sub-discipline Hydraulic Engineering, which is the study of the flow and transport of fluids (principally water and sewage). Hydraulic Engineers study the design of bridges, dams, channels, canals, levees, pipelines, irrigation systems and sewage treatment plants. The hydraulic engineer is also concerned with the transport of sediment in rivers, estuaries and at sea, and with the interaction between water and the sediment bed, which all lead to morphological changes, in coastal areas in particular.

The study of fluids is also of primary importance in other branches of civil engineering, such as construction (drying of concrete), soil mechanics (foundations of buildings in wet environments), and geoscience (groundwater flow).

The present course is called "Fluid Mechanics" as both fluid statics and fluid dynamics will be studied. This course is an introduction to the field, with an emphasis on civil engineering problems. This course therefore also covers aspects of hydraulic engineering or hydraulics, which is concerned with the study of the mechanical properties of liquids and their use in engineering practice.

In order to follow this course, students should be familiar with mathematical analysis (differentiation, integration, analytic functions, vectors) and dynamics (kinematics, momentum, force, work, energy).

Typical problems that will be studied in this course are illustrated in Fig. 1.

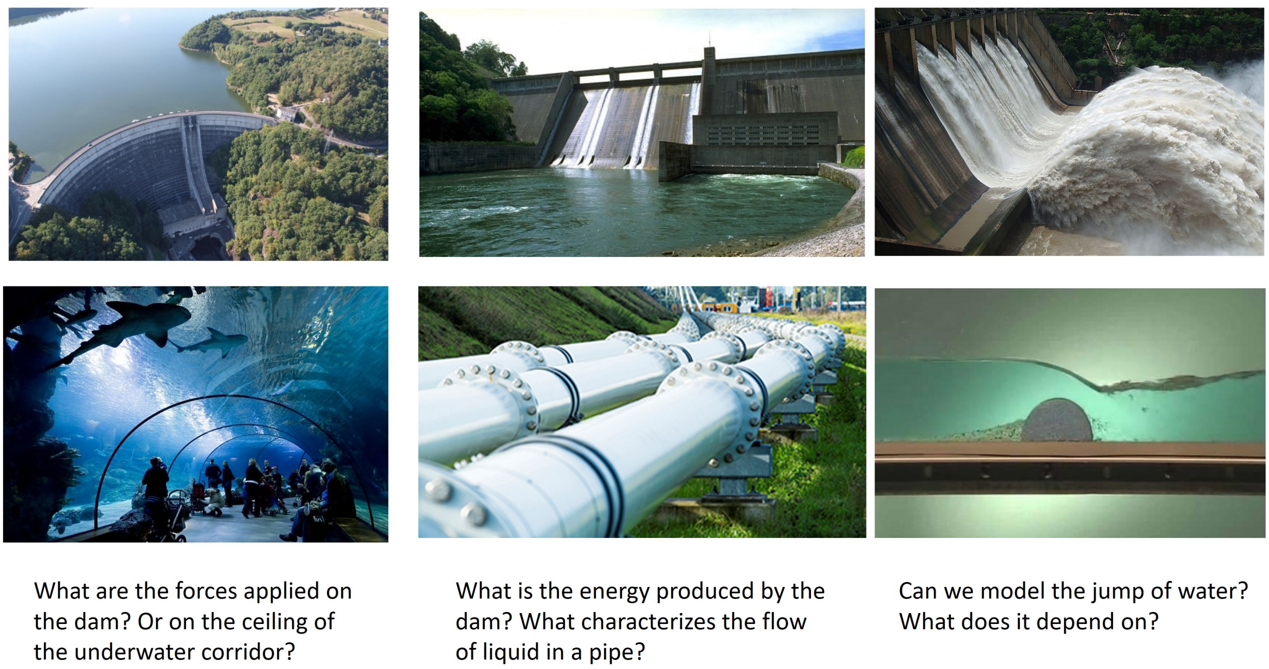


Figure 1: Examples of problems studied in this course.

0.2 Acknowledgements

The authors are grateful to Prof. em. dr. ir. J. Battjes and Dr. ir. R. J. Labeur, who prepared previous lecture notes on which these lecture notes are based.

1 Properties of fluids

1.1 What is a fluid?

Fluids are a phase of matter and include **liquids, gases** and plasmas. Plasmas will not be discussed in this course. The differences between solids, liquids and gases are illustrated in Fig. 1.1. A fluid is any material that cannot sustain a tangential, or shearing, force when at rest and that undergoes a continuous change in shape when subjected to such a stress.

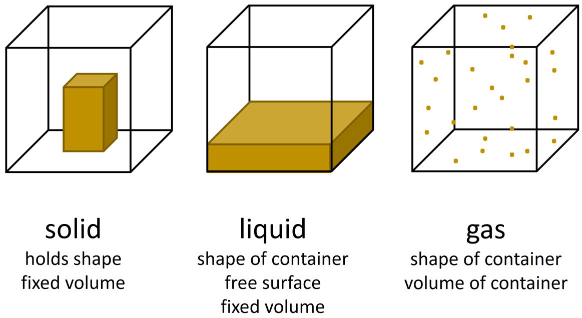


Figure 1.1: *Differences between solids, liquids and gases: a solid holds its shape and volume, a liquid takes the shape of the container and keeps a constant volume, whereas a gas will take the shape of and fill the whole volume of the container.*

In this course, we will mainly be discussing the properties of water (but not exclusively, as we will sometimes take oil, muddy water or air as examples). "Water" is the name of the liquid state of H_2O at standard conditions for temperature and pressure. It can form large drops (rain) and very small drops (fog, clouds). One may wonder whether fog is a gas: it is not because fog is a mixture of suspended microscopic droplets of *liquid* water in a *gas* (air). The technical name for fog is an **aerosol**. The gaseous state of water is **steam** or **water vapour**, and it is composed of free water molecules, forming a gas – usually mixed with air (another gas).

1.1.1 What is the difference between a liquid and a gas?

In the solid state, atoms remain fixed relative to one another (except for some small vibrations). In the gaseous state, atoms (or molecules) are moving around freely, which is why gas can expand as particles are getting further and further apart from one another. But what is a liquid – can it be seen as a very dense gas? Not completely: the main difference between a liquid and a gas is that a liquid keeps a constant volume (it will not expand on its own). One litre of water in a bucket will remain one litre of water, except if **evaporation** occurs. Evaporation is the phase change from liquid to gas: the part of liquid that becomes a gas will not keep a constant volume and expand (mix with the air). This phenomenon is typically studied in thermodynamics. The study of phase changes (evaporation, condensation, sublimation, etc.) is outside the scope of this course.

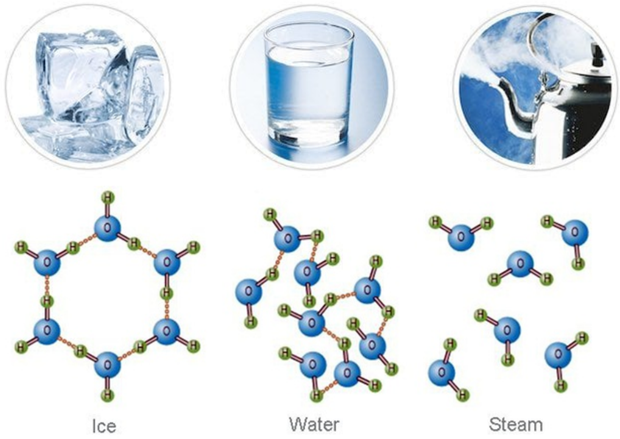


Figure 1.2: *Depending on the arrangement of water molecules, H_2O can form a solid, a liquid or a gas.*

Figure 1.3: *Two examples of the action of surface tension: (left) depending on its wetting properties, a droplet of water will spread more or less, forming a different contact angle with the underlying substrate; (middle and right) when a very thin tube is placed in a cup of water, water will spontaneously rise in the tube. The same holds in a larger tube filled with soil or in a concrete structure affected by humidity.*

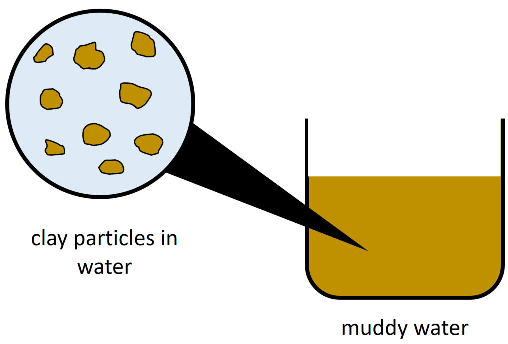


Figure 1.4: *A matter of scale: at microscopic scale muddy water can be seen as a suspension of individual clay particles in water, whereas at macroscopic scale muddy water is a fluid with bulk properties (density, viscosity).*

1.1.2 What scale are we studying?

We have been referring to water molecules (H_2O) and microscopic droplets. From a microscopic viewpoint, the study of the physics of fluids can be considered as a branch of thermodynamics. Do we care about microscopic viewpoints or thermodynamics in civil engineering? Yes, we sometimes do: only from a microscopic viewpoint can we understand capillary rise in tubes and wetting properties for instance – very important properties in many civil engineering applications (see Fig. 1.3). Even though these will not be studied in the course, it is important to know they exist.

In this course fluids will be studied from a **macroscopic** viewpoint. The smallest size we will use in this course is an element of volume that should be larger than, say, 1 mm^3. At this scale, water molecules or clay particles are not visible, and the fluid (for instance muddy water, see Fig. 1.4) can be considered a **continuum**, with which are associated important fluid properties such as **density** and **viscosity**, which will be defined and used in this course.

1.2 Bulk properties of fluids

The two important fluid properties that will be used in this course are density and viscosity.

1.2.1 Definition of density

The density of a fluid (but also of a solid!) is defined as the ratio of the mass of a fluid and its volume:

$$\rho = \frac{\text{mass}}{\text{volume}}. \tag{1.1}$$

The symbol for density is usually given by the Greek letter rho: ρ. The mass of 1 m^3 of water is 1000 kg. Therefore:

$$\rho_{\text{water}} = 1000 \text{ kg/m}^3. \tag{1.2}$$

The density of sea water is higher:

$$\rho_{\text{sea water}} = 1025 \text{ kg/m}^3. \tag{1.3}$$

This has as consequence that in estuaries fresh river water is found on top of sea water, as illustrated in Fig. 1.5.

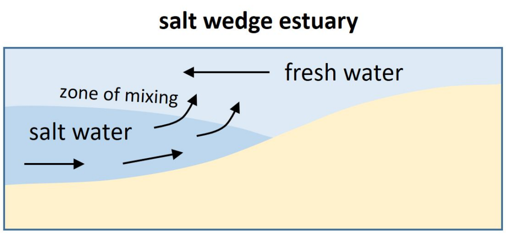


Figure 1.5: *When a river meets an estuary, the less dense fresh water is found on top of the denser sea water. Depending on the hydrodynamics, mixing is possible.*

The density of air is much smaller than the one of water:

$$\rho_{\text{air}} = 1.3 \text{ kg/m}^3. \tag{1.4}$$

The density of a fluid depends on its temperature: by increasing the temperature, the atoms are getting more agitated, and (for the same pressure) they will be further apart, hence the density of the fluid will be decreasing.

In this course we will assume that the densities of the liquids studied are constant. When the density of a fluid is able to change (for instance under the influence of the pressure), the fluid is said to be compressible. This is the case for gases (like air). By applying a pressure, one can force the volume of a confined fluid to decrease, while its mass remains the same. We will assume that the liquid most studied in this course (water) is not compressible. This implies that its density is constant.

The density of mixtures

Mixtures are quite common. A mixture of water and clay is called a clay suspension (as the clay particles are suspended in water), see Fig. 1.6. The density of a clay suspension is derived as follows:

$$\rho_{\text{fluid}} = \frac{\text{mass water} + \text{mass particles}}{\text{volume fluid}}. \tag{1.5}$$

Using the definition of density:

$$\begin{aligned} \text{mass water} &= \rho_{\text{water}} \times \text{volume occupied by the water}, \\ \text{mass particles} &= \rho_{\text{clay}} \times \text{volume occupied by particles}. \end{aligned} \tag{1.6}$$

Realizing that the volume fraction of particles is defined as

$$\phi = \frac{\text{volume occupied by particles}}{\text{volume fluid}} \tag{1.7}$$

and that

$$\text{volume occupied by particles} = \text{volume fluid} - \text{volume occupied by water}, \tag{1.8}$$

one obtains

$$\rho_{\text{fluid}} = (1-\phi) \times \rho_{\text{water}} + \phi \times \rho_{\text{clay}}. \tag{1.9}$$

One can verify that, if there are no suspended particles, $\phi = 0$ and $\rho_{\text{fluid}} = \rho_{\text{water}}$, whereas if there is no water $\rho_{\text{fluid}} = \rho_{\text{clay}}$. The density of clay is of the order of $\rho_{\text{clay}} = 2650\ \text{kg/m}^3$.

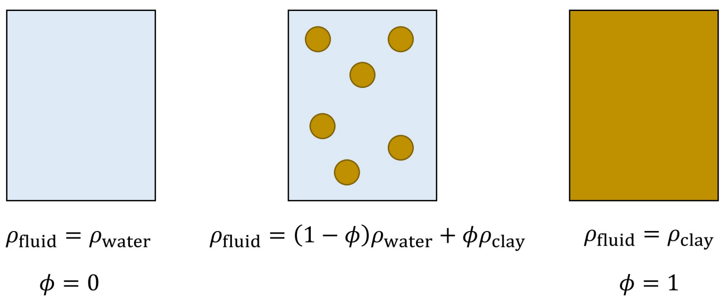


Figure 1.6: *Change in fluid density from left to right: a container filled with water, a container filled with a clay suspension and a container filled with clay.*

1.3 Definition of viscosity

The **viscosity** of a fluid is a measure of its resistance to deformation at a given shear rate. Viscosity originates from the friction between adjacent fluid layers that are in relative motion.

A fluid that has no resistance to deformation is called an **ideal** (or **inviscid**) **fluid**. This type of fluid has a zero viscosity and is observed only at very low temperature in so-called superfluids. Having a zero viscosity gives unconventional properties to superfluids: for instance, when stirred, superfluids form vortices that continue to rotate indefinitely! (Why? Because zero viscosity = zero friction = zero loss of energy). Water is not a superfluid, and it has a viscosity. The viscosity of water is about 10^{-3} Pa·s. Like density, the viscosity of a fluid is temperature dependent (see Fig. 1.7), as it depends on how the molecules of the fluid interact. What viscosity is can only be understood precisely by learning about **rheology**, which will be discussed in the next section.

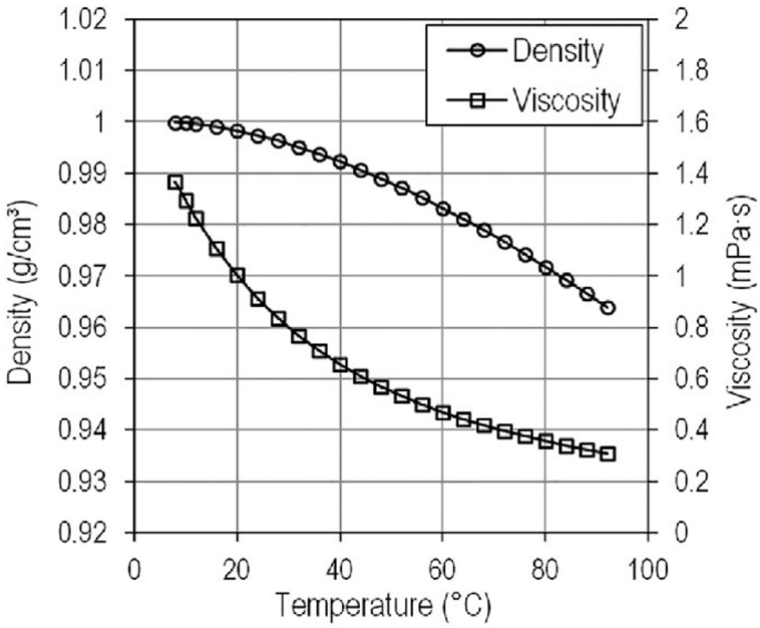


Figure 1.7: *Change in density and viscosity of fresh water as a function of temperature.*

1.3.1 Density or viscosity?

Density and viscosity are different properties, even though they can be correlated. For instance, until recently, the nautical bottom in harbours (which indicates the depth at which can be navigated) has been defined in terms of density. At the bottom of a harbour, there is a transitional depth region where sediment accumulates, and the density changes from water density to mud density. In order to be navigable, the fluid mud layer should have a density less than 1200 kg/m^3 (Port of Rotterdam, see Fig. 1.8). However, whether a vessel can navigate through fluid mud depends primarily on the flow behaviour of the fluid mud, and hence, on its viscosity.

A relationship exists between density and viscosity, but this relationship is different for every location in a harbour and for different harbours. Traditionally, a density of 1200 kg/m^3 has been used, as this is a very conservative estimate and a good criterion for every location. Today, this criterion is being revised by port authorities based on rheological experiments, with the hope of being able to deepen the nautical bottom, as this would save the port huge dredging costs.

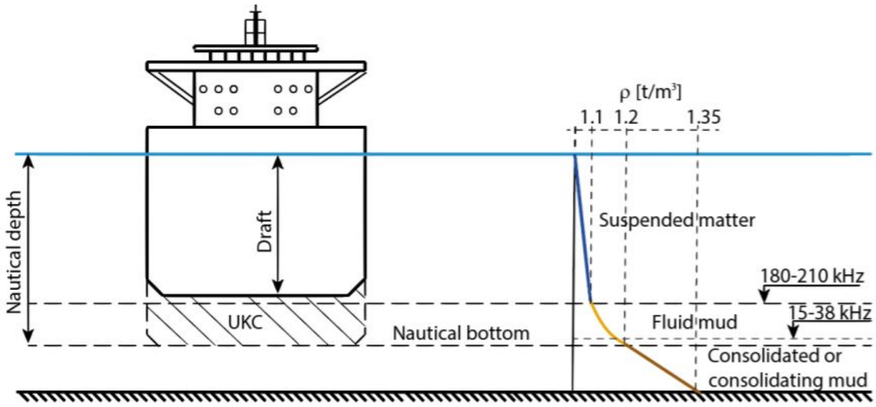


Figure 1.8: *In order to ensure safe navigation, a vessel has to respect the appropriate Under Keel Clearance (UKC) which can only be properly defined if the nautical bottom is known. The nautical bottom indicates the level beyond which contact with a ship's keel causes unacceptable effects on manoeuvrability. The density of the mud suspension is often assessed by high-frequency echo-sounder. The frequency domains associated with mud of different densities are indicated in the figure.*

1.4 Rheology

Rheology is the study of the flow of a liquid in response to an applied force (the word comes from the Greek root "rhei" , which means to flow). Here, we will illustrate how important variables (force, stress, shear rate and viscosity) are linked to each other.

In the illustration in Fig. 1.9, a liquid (in blue) is positioned between two flat plates (in grey). These plates have a surface are S (in m^2). One plate is fixed (the bottom plate), and the other one is pulled with a constant force F (in N). The pulling generates a displacement of the liquid in between the plates. For simplicity, we represent the fluid as slices, but one should keep in mind that these slices are very very small. All particles in a slice move with the same velocity, but each slice moves with a different velocity. The fact that the slices move with different velocities generates friction between the slices. It is this friction that is at the origin of viscosity.

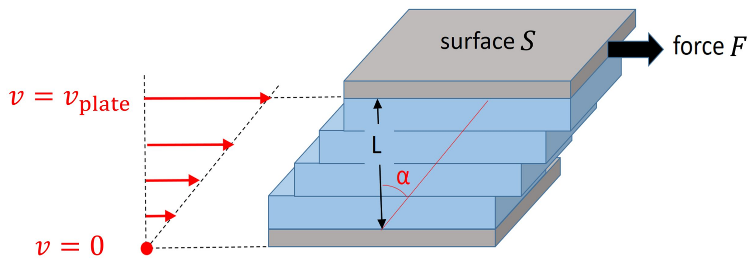


Figure 1.9: *Flow of liquid between a sliding and a fixed plate separated by a distance L. The velocity of each slice of liquid is represented by a different velocity (in red).*

The first important variable we will introduce here is **stress**. A stress has the same dimension as a pressure (Pa). A **pressure** is a stress that is defined as the result of a force *perpendicular* to the surface it is applied to. A stress can be applied in any direction to the surface. In the illustration in Fig. 1.9, you can see that the force F is applied *parallel* to the surface. The difference between stress and pressure is illustrated in Fig. 1.10.

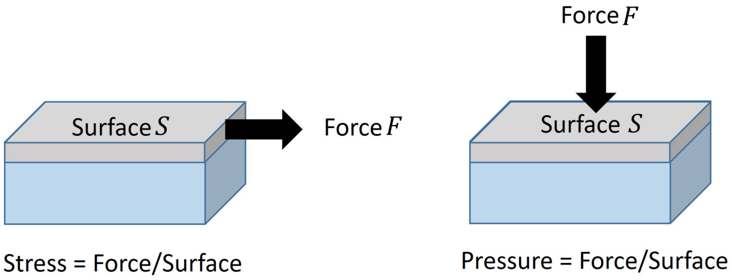


Figure 1.10: *Difference between a stress and a pressure: a pressure is a stress whereby the force is applied perpendicular to the surface it acts on.*

A stress is defined by

$$\tau = \frac{F}{S}, \tag{1.10}$$

where τ (in Pa $= \mathrm{N/m^2}$) is the stress, F (in N) is the force applied, and S (in $\mathrm{m^2}$) is the surface area.

We now need to define the velocity of the liquid in between the plates. To do this, we consider a small slice of fluid in the plane perpendicular to the plates shown in Fig. 1.11.

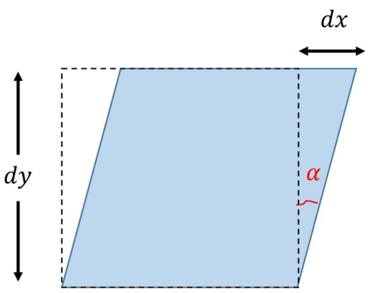


Figure 1.11: *Section of the liquid between the two plates. When the upper plate is moving, each slice of liquid is deformed.*

The deformation of the slice of liquid (of thickness dy) can be related to the angle α by:

$$\gamma = \frac{dx}{dy} = \tan(\alpha), \tag{1.11}$$

where γ is the shear strain, which is defined as the displacement dx of the top of the slice with respect to its bottom relative to the thickness dy of the slice. The shear rate of the fluid is defined as

$$\dot{\gamma} = \frac{d\gamma}{dt} = \frac{d\tan(\alpha)}{dt} = \frac{d}{dy}\frac{dx}{dt} = \frac{dv_x}{dy}, \tag{1.12}$$

where $\dot{\gamma}$ is the shear rate (in s^{-1}) and v_x (in m/s) is the velocity at which the top of slice is moving relative to its bottom. We deduce that

$$dv_x = \dot{\gamma} dy. \tag{1.13}$$

Note that the shear rate can be written as

$$\dot{\gamma} = \frac{dv_x}{dy}. \tag{1.14}$$

This equation can be integrated between the two plates:

$$\int_0^{v_x} dv_x = \int_0^y \dot{\gamma} dy. \tag{1.15}$$

After integration, we obtain for a constant shear rate $\dot{\gamma}$

$$v_x = \dot{\gamma} y. \tag{1.16}$$

We have used as boundary conditions:

$$v_x \left(y = 0 \right) = 0, \tag{1.17}$$

as the bottom plate is not moving, and

$$v_x \left(y = L \right) = v_{\text{plate}} = \dot{\gamma} L, \tag{1.18}$$

where v_{plate} is the velocity of the top plate, and L the separation between the plates.

The **viscosity** (also called the **shear or dynamic viscosity**) of the liquid is defined by

$$\tau = \frac{F}{S} = \eta \dot{\gamma} = \eta \frac{dv_x}{dy}. \tag{1.19}$$

The viscosity of a liquid η is the coefficient of proportionality between shear stress and shear rate.

1.5 Definitions of dynamic and kinematic viscosities

We have seen that the dynamic viscosity η is defined by

$$\tau = \eta \dot{\gamma}, \tag{1.20}$$

where η is the viscosity (in Pa·s), τ the shear stress (in Pa) and $\dot{\gamma}$ the shear rate (in s^{-1}). This equation means that if a shear rate $\dot{\gamma}$ is imposed on the liquid, the liquid will react with a shear stress (force per unit area). Vice-versa, if one imposes a shear stress τ on the liquid, it will flow with a shear rate $\dot{\gamma}$. These two actions can be studied using a device called a rheometer, as illustrated in Fig. 1.12. The most standard rheometers are cylindrical, as the liquid can then be constrained between the outer cylinder and the inner cylinder. The inner cylinder is the one rotating.

A second measure of viscosity that is often used in fluid mechanics is the kinematic viscosity, defined as

$$\nu = \eta / \rho, \tag{1.21}$$

where ν is the kinematic viscosity (in m^2/s) and ρ the density of the liquid (in kg/m^3).

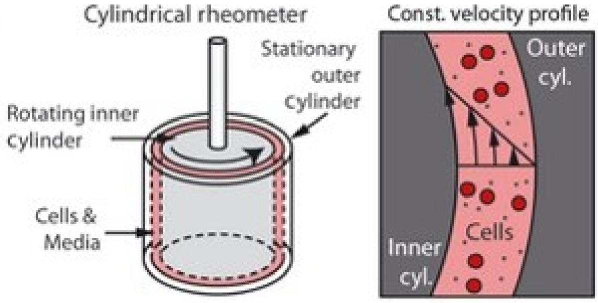


Figure 1.12: *Schematic representation of a cylindrical rheometer. The gap between the two cylinders is very small to ensure a linear velocity profile between the two cylinders. Here the rheology of blood is analysed.*

1.6 Force on a volume element fluid

For the rest of the course, it is important to be able to evaluate the friction force on a volume element of liquid. The force we are going to derive now is one of the very important forces that is used in the famous Navier–Stokes equation, which describes the hydrodynamics of water bodies and is implemented in large-scale numerical models, such as Delft3D, Telemac or MIKE.

In order to derive this force, we will use the definitions and derivations in the previous section. In particular, as shown in Fig. 1.13, we can now define the force per unit area on the top and bottom parts of a slice of liquid (symbolized by the grey areas in the figures).

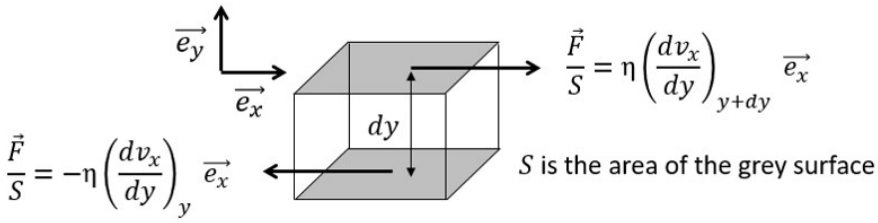


Figure 1.13: *Forces on the top and bottom of a slice of liquid.*

The total friction force $d\vec{F}_{\text{friction}}$ on a slice of fluid of thickness dy is the sum of the force on the top and the bottom:

$$\frac{d\vec{F}_{\text{friction}}}{S} = \eta \left(\frac{dv_x}{dy}\right)_{y+dy} \vec{e_x} - \eta \left(\frac{dv_x}{dy}\right)_{y} \vec{e_x}. \tag{1.22}$$

where $\vec{e_x}$ is the unit normal vector in the direction of the flow. Note that the minus sign arises from the fact that we work with vectors, and that the orientations of the forces are opposite to each other. The equation can be rewritten:

$$d\vec{F}_{\text{friction}} = \eta \frac{d^2 v_x}{dy^2} S dy \vec{e_x}. \tag{1.23}$$

1.7 Complex fluids

Some fluids resist flow in a complex way, meaning that the shear stress they experience does not just depend linearly on the strain, as in the previous section. Examples of complex fluids are mud suspensions that are found in harbours and coastal regions, as illustrated in Fig. 1.14. The study of these fluids is outside of the scope of the present course.

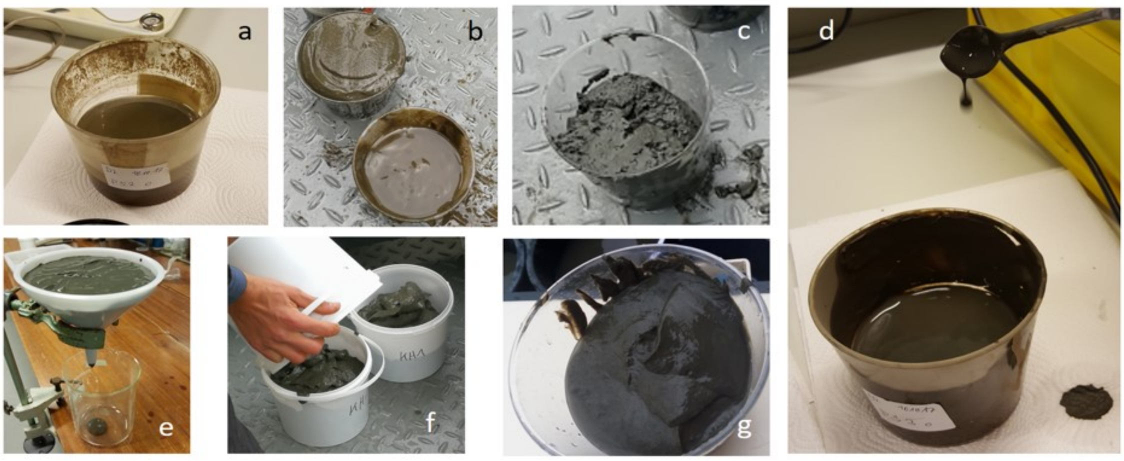


Figure 1.14: *Examples of visco-elastic fluids found in harbours.*

For example, there can be a partial elastic response of the fluid. Such fluids are called visco-elastic fluids. A pure elastic solid obeys Hooke's law:

$$\tau = G\gamma, \tag{1.24}$$

where G is the shear modulus. A visco-elastic fluid obeys the so-called Maxwell model:

$$\dot{\gamma} = \frac{\tau}{\eta} + \frac{\dot{\tau}}{G}, \tag{1.25}$$

which is a combination of a pure viscous and pure elastic response.

1.8 Laminar and turbulent flows

When one studies the rheology of fluids (with a rheometer) a necessary condition is that the flow is laminar. This implies that there is no mixing between the different layers (slices or "laminae") of fluid. An obstruction or high fluid velocities generate turbulence, which mixes the fluid, as shown in Fig. 1.15.

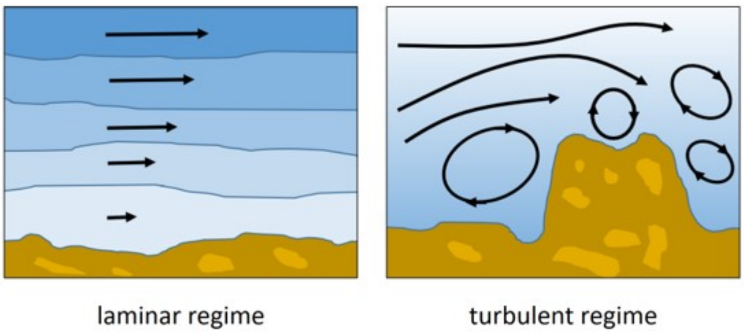


Figure 1.15: *Differences between laminar and turbulent flows.*

Whether a flow is laminar can be evaluated by estimating the Reynolds number, Re, which represents the ratio of inertial forces (which create turbulence and mixing) to viscous forces (which create friction and act the retain laminarity):

$$\mathrm{Re} = \frac{v\rho L}{\eta} = \frac{vL}{\nu}, \tag{1.26}$$

where v is the fluid's velocity, L is a characteristic length (for instance the size of the obstruction), and the kinematic viscosity ν is given by η/ρ, where η is the dynamic viscosity and ρ the density of the fluid. The number was invented by George Gabriel Stokes (1819-1903) but is named after Osborne Reynolds (1842-1912), who popularised its use. Laminar flows occur at low fluid velocities, or low Reynolds numbers. The Reynolds number will be discussed in more detail in Chapters 3 and 6.

1.9 Summary

After studying this Chapter you should:

1. Be able to tell the differences between a solid, a liquid and a gas.
2. Be able to define the density of a fluid and a mixture.
3. Be able to define an ideal (inviscid) fluid.
4. Understand the differences between a stress and a pressure and define both.
5. Be able to define a shear strain.
6. Be able to define the fluid velocity as function of shear rate.
7. Be able to define the viscosity as function of (1) shear stress and shear rate and (2) as a function of the derivative of the fluid velocity in the across-stream direction.
8. Be able to explain how we can measure viscosity experimentally.
9. Be able to define the kinematic viscosity.
10. Be able to define the force on a volume element of fluid.
11. Be able to define a visco-elastic fluid.
12. Know the difference between a laminar and a turbulent flow

The Dutch corner

In het Nederlands is er geen word voor "fluid"... of toch wel! Het is **fluïdum**. Een vloeistof is een "liquid".

English	Nederlands
ideal (inviscid) fluid	ideale vloeistof
rheology	reologie
density	dichtheid
viscosity	viscositeit
shear stress	schuifspanning
shear rate	afschuifsnelheid
strain	vervorming
shear modulus	schuifmodulus

Een ideale vloeistof is een incompressibele vloeistof zonder inwendige wrijving. Ideale vloeistoffen ondervinden geen schuifspanningen, viscositeit en warmteoverdracht.

2 Hydrostatics

2.1 Introduction

Hydrostatics is the study of fluid statics, i.e., the study of fluids at rest as opposed to fluid dynamics, the study of fluids in motion. When fluids are at rest they cannot be defined by their viscosity (as viscosity is a property of fluids in motion), but only their density. In this Chapter we are going to review how to evaluate a pressure in a fluid at rest, the so-called **hydrostatic pressure**, at any position in this fluid and how to calculate the associated pressure force on an element volume. We will define what a piezometric (or hydraulic) head is. We will discuss the principle of communicating vessels, Pascal's law and Torricelli's experiment. Finally, we will study Archimedes' principle.

2.2 Pressure forces on an element of fluid

We start by considering a small element (cube) of fluid at rest inside a larger volume of the same fluid also at rest, as shown in Fig. 2.1.

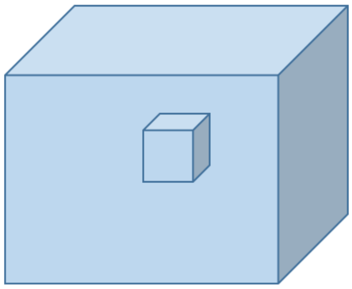


Figure 2.1: *We consider a small fluid cube inside a large volume of fluid at rest.*

We are going to evaluate the forces exerted on the small element of fluid. Even though the fluid is at rest, its molecules are continuously moving. By moving, they will collide into the sidewalls of the cube and create a pressure. The fluid feels the pressure on all sides. We will first derive the pressure forces on the sides perpendicular to the horizontal axis, as shown in Fig. 2.2.

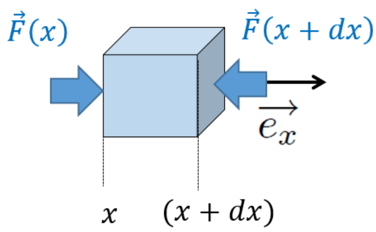


Figure 2.2: *We apply horizontal pressure forces to the fluid cube.*

The net force is expressed in vector form as

$$d\vec{F} = \vec{F}(x + dx) + \vec{F}(x), \tag{2.1}$$

where the lower-case d in $d\vec{F}$ reflects the fact that we are considering a small force (over the small distance dx). Projecting the vector $\vec{F} = (F_x, F_y, F_z)$ along the x-axis in the direction of the unit vector $\vec{e_x}$ one obtains in scalar form

$$dF_x = -F_x(x + dx) + F_x(x). \tag{2.2}$$

Recalling the definition of a pressure, one obtains

$$\begin{aligned} dF_x &= -P(x + dx)S + P(x)S, \\ &= -\left[P(x + dx) - P(x)\right]S, \end{aligned} \tag{2.3}$$

where S is the area of a side of the cube. It follows that

$$dF_x = -\frac{\partial P(x)}{\partial x} S dx. \tag{2.4}$$

If the pressure on both sides is the same, $P(x + dx) = P(x)$ and therefore $dF_x = 0$: when there is no pressure difference, the fluid is at rest, as is the case here. The above derivation can be repeated for the $\vec{e_y}$-direction, replacing x by y throughout. This pressure force will be used to derive the Navier–Stokes equations in a later Chapter. In the next section, we will see what happens in the vertical direction.

We conclude that in a fluid at rest, the pressure at any position in any horizontal plane is equal. If that were not the case, a force would act, and the fluid would start to move.

We can use vector notation to generalize (2.4):

$$d\vec{F} = -\left(\frac{\partial P}{\partial x}\vec{e_x} + \frac{\partial P}{\partial y}\vec{e_y} + \frac{\partial P}{\partial z}\vec{e_z}\right) dV, \tag{2.5}$$

where we are now considering a small cube of size $dx \times dy \times dz$, so that the relevant surface areas on which the pressure acts in the x, y, and z-directions are $dydz$, $dxdz$, and $dxdy$, respectively. The volume of the small cube is $dV = dxdydz$. Note that we need to use partial derivatives in (2.5).

We can express (2.5) in terms of the pressure force per unit volume $\vec{f} = d\vec{F}/dV$ (N/m^3), which is given by

$$\vec{f} = -\vec{\nabla} P, \tag{2.6}$$

where $\vec{\nabla}$ denotes the gradient. We recall that, in Cartesian coordinates,

$$\vec{\nabla} P = \frac{\partial P}{\partial x}\vec{e_x} + \frac{\partial P}{\partial y}\vec{e_y} + \frac{\partial P}{\partial z}\vec{e_z}. \tag{2.7}$$

2.3 Vertical force on an element fluid

We will now derive the pressure exerted in the vertical direction on a small fluid element. In this case, we have to account for a very important force that plays a role even when the fluid is at rest: the force linked to gravity.

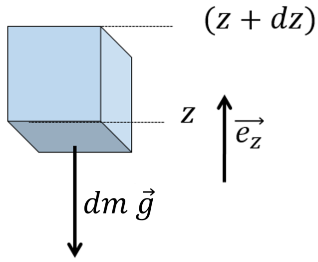


Figure 2.3: *We apply vertical pressure forces to the fluid cube and we do not forget its (small) weight related to its (small) mass dm.*

To set up the force balance, we use Newton's second law applied to the small fluid element:

$$dm\ \vec{a} = -dV\ \vec{\nabla}P + dm\ \vec{g}, \tag{2.8}$$

where dm is the mass of the small cube, dV its volume and g the gravitational acceleration. The mass of the fluid element and its density are linked by:

$$\rho = \frac{dm}{dV}. \tag{2.9}$$

Dividing the equation by the volume of the small element, we get

$$\rho\vec{a} = -\vec{\nabla}P + \rho\vec{g}. \tag{2.10}$$

As the fluid is at rest, its acceleration is zero: $\vec{a} = \vec{0}$. If we project the equation along the x, y and z-directions, we obtain:

$$\begin{aligned} 0 &= -\frac{\partial P}{\partial x}, \\ 0 &= -\frac{\partial P}{\partial y}, \\ 0 &= -\frac{\partial P}{\partial z} - \rho g. \end{aligned} \tag{2.11}$$

The first two equations indicate that the pressure does not change in the x and y-directions, as we learned is the case for a fluid at rest in the previous section.

The final equation indicates that, in the presence of gravity, the pressure in a fluid at rest is a function of the vertical coordinate z, known as the hydrostatic pressure.

The dependence of pressure on the vertical coordinate z arises because of gravity. In zero gravity, one finds (as for the horizontal planes) that the pressure is the same at any height, i.e., $P(z) = P(z + dz)$ for any z. In zero gravity, the pressure in a fluid is therefore the same everywhere. This is why it is very easy for a liquid to get out of a glass in a space station.
We are now going to apply this equation to the situation of a water reservoir with a water height h, as shown in Fig. 2.4. Integrating (2.11), we get

$$\int_{P_{\text{atm}}}^{P(z)} dP = -\rho g \int_h^z dz. \tag{2.12}$$

Here, we have used the assumption that the density of the fluid ρ is constant and can be taken out of the integral. The integral can be solved to give:

$$P(z) = P_{\text{atm}} + \rho g\,(h - z)\,, \tag{2.13}$$

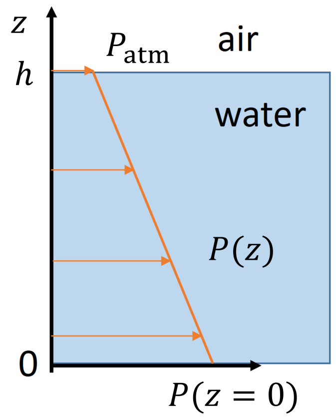


Figure 2.4: *Pressure variation in an open water reservoir of height h. The arrows represents the pressure forces (pressure multiplied by an unit of surface) at different depths. For the same unit of surface, the force increases with depth, as does the pressure.*

where P_{atm} is the atmospheric pressure acting at the top of the open water reservoir. The pressure $P(z) - P_{\text{atm}} = \rho g\,(h-z)$ is known as the **hydrostatic pressure**. The hydrostatic pressure decreases with z. In other words, it becomes larger the further we move down in the water column.

Equation (2.13) can also be rewritten as

$$h = z + \frac{P(z) - P_{\text{atm}}}{\rho g}, \tag{2.14}$$

where h is called the **hydraulic head** (or **piezometric head**). The hydraulic head is the sum of the vertical position z and the normalized pressure, given here relative to atmospheric pressure. The hydraulic head is a convenient way to define the pressure in the water column, as, in contrast to $P(z)$ which depends on the height z in the water column, the hydraulic head h is the same at any height in the water column. This is illustrated in Fig. 2.5. The piezometric head can be defined relative to the bottom of a vessel or, more generally, relative to any fixed vertical position. We will see later in the course how the hydraulic head is linked to the mechanical energy of the fluid.

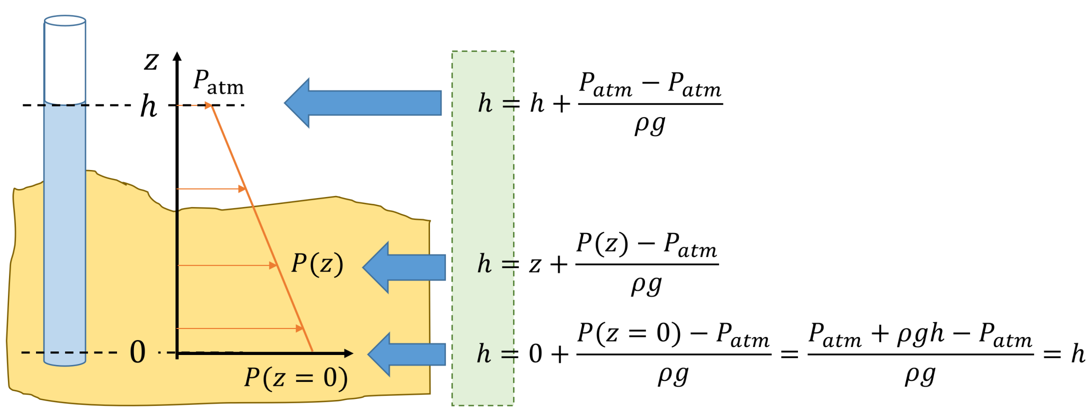


Figure 2.5: *Definition of the piezometric head: in a fluid at rest, the piezometric head is the same at any position in the fluid.*

We have seen that, in order to induce fluid motion, there should be a pressure difference. The opposite is also true: if there is no pressure difference, the fluid will remain at rest. This implies that sets of containers that are connected will always exchange their fluid (under the influence of a pressure difference) until equilibrium is reached (no pressure difference). This is the principle of **communicating vessels**, as illustrated in Fig. 2.6.

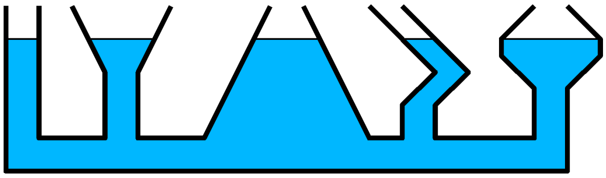


Figure 2.6: *Regardless of the shape and volume of the vessels, the liquid is at rest when the pressure is the same at any depth. This pressure can be represented by the piezometric head, which is equal to the distance between the fluid/air interface and the bottom of the vessel.*

The principle of communicating vessels is found in nature, in the form of Artesian aquifers, as shown in Fig. 2.7. The name originates from Artois a French province where this type of aquifers are found.

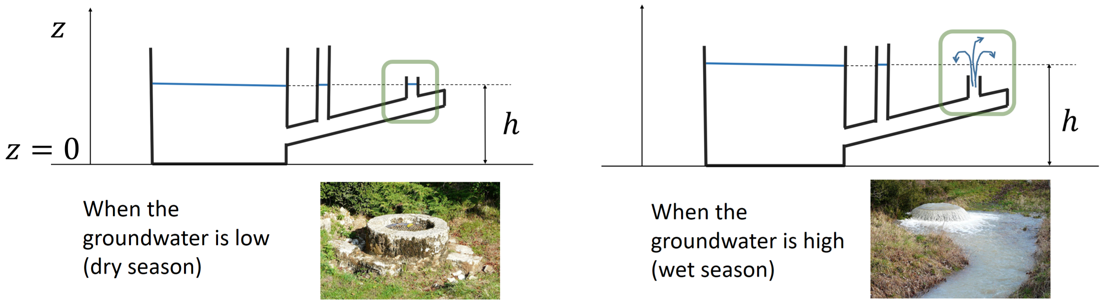


Figure 2.7: *The working principle of an Artesian aquifer: the water will emerge from the aquifer's opening on the right when the piezometric head h is higher than the height of the aquifer's opening.*

2.4 Pascal's law

Pascal's law (also called Pascal's principle or the principle of transmission of fluid pressure) is a principle that states that a pressure change at any point in a confined incompressible fluid is transmitted throughout the fluid such that the same change occurs everywhere. The law was established by French mathematician Blaise Pascal in 1653.

Pascal's barrel is the name of a hydrostatics experiment allegedly performed by Blaise Pascal in 1646. In the experiment, Pascal supposedly inserted a long vertical tube into a barrel filled with water. When water was poured into the vertical tube, the increase in hydrostatic pressure caused the barrel to burst.

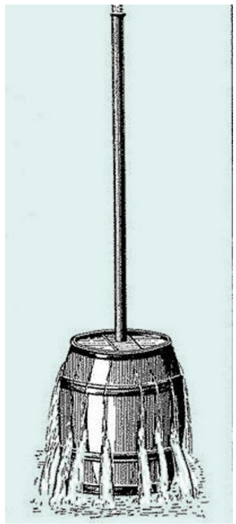


Figure 2.8: Pascal's barrel.

The experiment is mentioned nowhere in Pascal's preserved works and it may be apocryphal, attributed to him by 19th-century French authors, among whom the experiment is known as "crève-tonneau", which loosely translates as the "barrel-buster"; nevertheless the experiment remains associated with Pascal in many elementary physics textbooks.

So, why does the barrel bust? From the equation derived above, one can estimate that if the tube is long enough, the pressure at the bottom end of the tube (the end in contact with the barrel) will be very large:

$$P_{\text{top of barrel}} = P_{\text{atm}} + \rho g H, \tag{2.15}$$

where H is the height of the tube. Through Pascal's principle, this pressure will be equal to the pressure at any position on the inside surface of the barrel:

$$P_{\text{barrel}} = P_{\text{atm}} + \rho g H, \tag{2.16}$$

where we have neglected the small change in pressure between the top and bottom of the barrel, i.e., we say that $H \gg d$, where d is the height of the barrel. For example, if we take $H = 5$ m, we can estimate that $\rho g H = 1000 \times 9.81 \times 5 = 5 \times 10^4$ Pa. Is that a high pressure? The atmospheric pressure is $P_{\text{atm}} = 10^5$ Pa, so clearly $\rho g H \ll P_{\text{atm}}$. We now calculate the force on the surface of the barrel. We used cylindrical instead of Cartesian coordinates:

$$F = -\left[P(r + dr) - P\left(r\right)\right] S, \tag{2.17}$$

where $P\left(r + dr\right)$ is the pressure just outside the barrel of radius r, and $P\left(r\right)$ the pressure just inside. The surface area S is the surface area of the barrel, let's say 1 m^2. We know that the pressure outside of the barrel is

$$P(r + dr) = P_{\text{atm}}, \tag{2.18}$$

and the pressure inside the barrel is

$$P(r) = P_{\text{atm}} + \rho g H. \tag{2.19}$$

The total (net) force on the wall of the barrel is given by

$$F = -\left[P_{\text{atm}} - P_{\text{atm}} - \rho g H\right] S = \rho g H S. \tag{2.20}$$

The force F is positive and, as the unit vector $\vec{e_r}$ in cylindrical coordinates is directed from the inside to the outside of the barrel, we now know that if the barrel explodes, it will be associated with an outward force and thus and outward motion (from inside to outside). Note that the atmospheric pressure is eliminated from the equation. So, even though $\rho g H \ll P_{\text{atm}}$, it is still $\rho g H$ that matters in the calculation of the force!

We can calculate the force to be:

$$F = 5 \times 10^4 \text{ Pa} \times 1 \text{ m}^2 = 5 \times 10^4 \text{ N}. \tag{2.21}$$

To estimate whether this is a large force, we can express F as if F were produced by the weight of a mass m:

$$F = mg = 5 \times 10^4 \text{ N}. \tag{2.22}$$

To generate 5×10^4 N requires a mass of approximately 5×10^3 kg. Clearly, if the barrel is not strong enough, it will explode. It is said that an ancient Roman mining technique is based on this principle, as described by Pliny the Elder. The miners would excavate narrow cavities down into a mountain and fill the cavities with water, which would cause pressures large enough to fragment thick rock walls.

From this simple calculation, it is evident that to generate a net force, there should be a pressure gradient (a pressure difference across a certain distance divided by this distance). We also showed that even if the atmospheric pressure is very large compared to the pressures imposed on fluids or objects, the atmospheric pressure generally does not play a role in the estimation of forces, as atmospheric pressure acts everywhere on Earth, and only pressure differences matter. In an empty barrel, for instance, there would be atmospheric pressure on both sides of the barrel, and hence no net force would be applied:

$$F = -\left[P_{\text{atm}} - P_{\text{atm}}\right] S = 0. \tag{2.23}$$

Another experiment would be to create a vacuum in a closed vessel. In that case, one can estimate that (the pressure of a vacuum is close to zero):

$$F = -\left[P_{\text{atm}} - 0\right] S = -P_{\text{atm}} S. \tag{2.24}$$

If an accident occurs, it would be an implosion (not an explosion) of the vessel: the minus sign indicates that the force is directed towards the centre of the vessel. Note that after the initial implosion, the broken elements will collide in the centre, and this will lead to an explosion.

2.5 Pressure in closed vessels: example 1

We now consider the set-up shown in Fig. 2.9. If we were to drill a hole in the small closed part of the reservoir, which has a pressure difference with the outside of $\Delta P = P(z_1) - P(z_2) > 0$, water would start to drain out, and we would have the situation of an Artesian aquifer. The water (and the gas) in the small closed part of the reservoir has a pressure $P(z_1)$ that is higher than the atmospheric pressure $P_{\text{atm}} = P(z_2)$. In exercises, the atmospheric pressure is very often defined to be zero, as in Fig. 2.9. Setting the atmospheric pressure equal to zero ($P_{\text{atm}} = 0$) is just a convenience, as we have seen that the net pressure force only depends on the pressure difference (the atmospheric pressure, which acts everywhere, can be eliminated from the equation).

The pressure at position z_0 is defined by:

$$z_2 - z_0 = \frac{P(z_0) - P(z_2)}{\rho g} = \frac{P(z_0) - P_{\text{atm}}}{\rho g}. \tag{2.25}$$

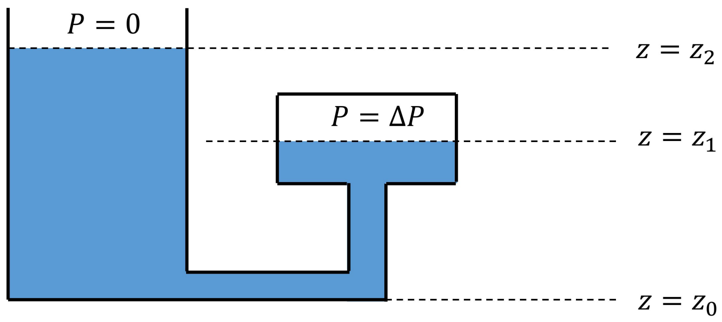


Figure 2.9: *One part of this reservoir is in contact with atmospheric pressure, another part is closed and has air at a different pressure.*

The pressure at position z_1 is given by:

$$z_1 - z_0 = \frac{P(z_0) - P(z_1)}{\rho g}. \tag{2.26}$$

By subtraction of the two previous equations, one can show that

$$z_2 - z_1 = \frac{P(z_1) - P_{\text{atm}}}{\rho g}. \tag{2.27}$$

This implies that we control the pressure in the closed part (i.e., $P(z_1)$) by changing the height difference $z_2 - z_1$. To give an order of magnitude, if we would like $\Delta P = P(z_1) - P_{\text{atm}} = 12$ kPa, then we need to impose

$$z_2 - z_1 = \frac{12 \times 10^3 \text{ Pa}}{1.0 \times 10^3 \text{ kg/m}^3 \times 9.81 \text{ m/s}^2} = 1.2 \text{ m}. \tag{2.28}$$

Note that $P_{\text{atm}} = P(z_2) < P(z_0)$, and $P(z_1) < P(z_0)$: the pressure at the lowest depth is always the highest. Similarly $P_{\text{atm}} = P(z_2) < P(z_1)$, as the position z_1 lies deeper than z_2.

The piezometric head is given by

$$h = z_2 - z_0 = (z - z_0) + \frac{P(z) - P_{\text{atm}}}{\rho g}. \tag{2.29}$$

One can verify that at any position z in the fluid, this equation is satisfied. In a fluid at rest, as we have discussed earlier, the piezometric head should be the same at any point in the fluid.

2.6 Pressure in closed vessels: example 2

We now consider the following situation: imagine that you fill a glass of water to the rim and cover it with a piece of strong paper. You then turn the glass around (holding the paper). When the glass is upside-down, you remove your hand from the paper. You will observe that the paper remains attached to the glass. Why?

To explain what happens, we have to study the force balance on the piece of paper. It is clear that there is a force equilibrium, otherwise the paper would fall.

The pressure on the side where the paper is in contact with the air is the atmospheric pressure P_{atm}, and therefore the force balance on the paper gives:

$$\vec{F_1} + \vec{F_2} = F_1 \vec{e_z} - F_2 \vec{e_z} = P_{\text{atm}} S \vec{e_z} - (P_2 S + mg)\, \vec{e_z}, \tag{2.30}$$

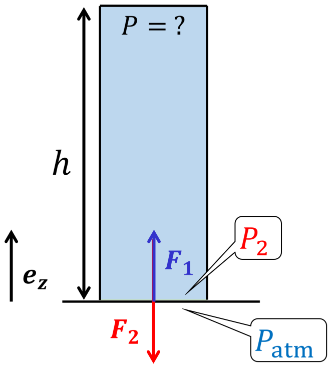
 

Figure 2.10: *Forces exerted on a piece of paper located at the rim of a glass of water turned upside-down.*

where S is the surface area of the paper touching the water, and m is the paper's mass. For all the parts of the paper not touching the water, there is atmospheric pressure on both sides of the paper and hence no net pressure force. The weight of the paper is given by mg. If we were to use very heavy paper, it is clear that the paper would fall, as mg would be larger than all the other forces. We will neglect the weight of the paper, as we assume it is small compared to the other forces. We also neglect another force: the capillary suction force between the wet part of the glass and the paper. Neglecting these forces and imposing force equilibrium, i.e., $\vec{F}_1 + \vec{F}_2 = \vec{0}$, we obtain

$$P_{\text{atm}} = P_2. \tag{2.31}$$

Therefore, the pressure on the side of the paper that is contact with the water is the atmospheric pressure. From the the relationships for hydrostatic pressure, we know that the pressure at the (upside-down) bottom of the glass P should be lower than $P_2 = P_{\text{atm}}$. To calculate P we apply the equation derived above:

$$\int_{P_{\text{atm}}}^{P} dP = -\rho g \int_0^h dz. \tag{2.32}$$

Here we have used the assumption that the density ρ of the fluid is constant and can be taken out of the integral. The equation is solved to give:

$$P = P_{\text{atm}} - \rho g h. \tag{2.33}$$

We can verify that $P < P_{\text{atm}}$. Pressure is a scalar quantity that should always be positive or equal to zero (a zero pressure represents vacuum, when no molecules are present, and hence no molecule can create pressure forces by colliding with surfaces). Therefore we have to impose:

$$P_{\text{atm}} - \rho g h \geqslant 0, \tag{2.34}$$

which results in

$$h \leqslant \frac{P_{\text{atm}}}{\rho g}. \tag{2.35}$$

For water and standard atmospheric pressure, one can estimate that

$$h \leqslant \frac{10^5 \text{ Pa}}{10^3 \text{ kg/m}^3 \times 9.81 \text{ m/s}^2} = 9.81 \text{ m}. \tag{2.36}$$

This implies that if a closed upside-down cylinder of height greater than 10 m is used instead of a glass, the part above 10 m will be "filled" with vacuum.

2.7 Torricelli's experiment

The experiment discussed in the previous section (filling a more than 10 m tall column with water to show that a vacuum forms in the part of the column above 10 m) is difficult to realize in a laboratory. This why Torricelli (1609-1647), who was studying the problem, had the following idea: the maximum filling height depends on density, so it is to better use a liquid with very high density. He chose mercury, which has a density of 1.3545×10^4 kg/m^3. One can estimate that (using the average atmospheric pressure at sea level):

$$h \leqslant \frac{1.01325 \times 10^5 \text{ Pa}}{1.3545 \times 10^4 \text{ kg/m}^3 \times 9.81 \text{ m/s}^2} = 76 \text{ cm}. \tag{2.37}$$

Figure 2.11: *Torricelli (1609-1647).*

Torricelli could confirm that the mercury did indeed not rise more than 76 cm in the test tube he was using. As the measurement with mercury is very precise, he discovered a new application of the principle: the barometer. Indeed, the rise of the mercury could be correlated to a change in atmospheric pressure. During storms, the atmospheric pressure can drop to 9.8×10^4 Pa, and hence

$$h \leqslant \frac{9.8 \times 10^4 \text{ Pa}}{1.3545 \times 10^5 \text{ kg/m}^3 \times 9.81 \text{ m/s}^2} = 73 \text{ cm}. \tag{2.38}$$

In Fig. 2.11 you can see that Torricelli is placing a tube filled with mercury in a large vessel (also filled with mercury). The mercury drops from the tube, and a vacuum is created in the upper part of the tube until equilibrium is reached. The pressure of mercury at the point the tube is touching the mercury in the vessel is the atmospheric pressure (as the top of the mercury bath is in contact and in equilibrium with air).

2.8 Pressure forces on inclined surfaces – the integral method

So far, we have been studying pressure forces assuming that they are always in the direction perpendicular to the surfaces they are acting on. Is that always true? Yes.

Pressure forces originate from the collision of molecules of the fluid with the surface: this happens in a random fashion, so that on average the only direction in which the collision forces do not cancel out is the direction perpendicular to the surface. This is also why the pressure force in a vacuum is zero: there are no particles in a vacuum, so no collisions.

Let's now calculate the pressure force on an inclined wall, as illustrated in Fig. 2.13.

The unit vector perpendicular to the plate is given by

$$\vec{e_n} = \vec{e_x} \cos(\theta) - \vec{e_z} \sin(\theta) \,. \tag{2.39}$$

The pressure force on a small surface strip $dS = dl \times d_2$ at position z is given by

$$d\vec{F}(z) = P(z) d_2 dz / \cos(\theta) \vec{e_n}, \tag{2.40}$$

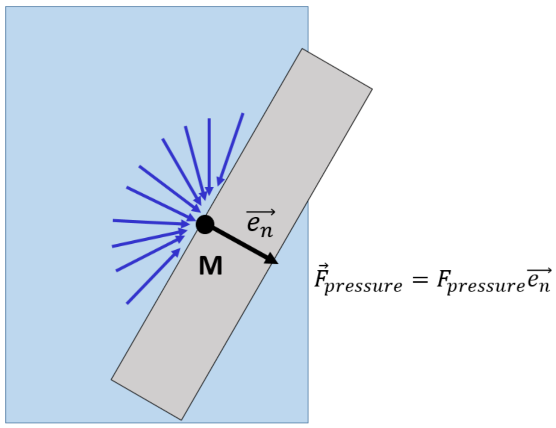


Figure 2.12: *The pressure force at point M is directed perpendicular to the surface at M. The molecules of fluid collide randomly with the surface at M, creating an average pressure force that is always directed perpendicular to the surface at M.*

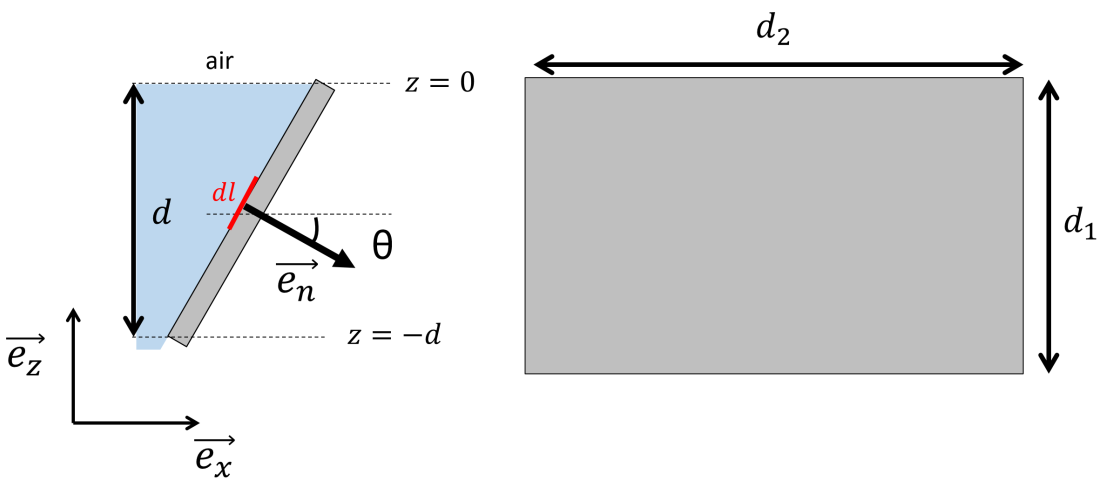


Figure 2.13: *An inclined plane is in contact with a fluid (in blue). The plate has a surface area $d_1 \times d_2$, and the length d_1 is related to d by $d = d_1 \cos(\theta)$.*

where we have used the fact that $dl = dz/\cos(\theta)$. The pressure is given by

$$P(z) = -\rho g z + P_{\text{atm}}. \tag{2.41}$$

Note that we have chosen $z = 0$ to be the fluid/air interface, so that $P(z = 0) = P_{\text{atm}}$. One could also choose another reference level (such as $z = 0$ at the bottom of the plate; in that case $P(z) = \rho g (d - z) + P_{\text{atm}}$), but the final result for the force will be the same.

On one side of the plate there is the fluid, and on the other there is air. Therefore, the total net force on the plate is expressed as:

$$\vec{F} = \int_{z=-d}^{0} \left[-\rho g z + P_{\text{atm}}\right] d_2 dz/\cos(\theta)\, \vec{e_n} - \int_{z=-d}^{0} P_{\text{atm}} d_2 dz/\cos(\theta)\, \vec{e_n}. \tag{2.42}$$

As we have already stated earlier, the contribution of the atmospheric pressure can be eliminated, as the resulting forces on both sides of the plate are equal and opposite. We obtain:

$$\vec{F} = -\int_{z=-d}^{0} \rho g z d_2 dz/\cos(\theta)\, \vec{e_n}. \tag{2.43}$$

We see that, as expected, the force is directed in the same direction as $\vec{e_n}$, as all the coefficients are positive. After integration, one obtains

$$\vec{F} = \rho g \frac{d^2 d_2}{2\cos(\theta)} \vec{e_n}. \tag{2.44}$$

After substituting for $\vec{e_n}$ one obtains

$$\vec{F} = \frac{1}{2} \rho g d^2 d_2 \left(\vec{e_x} - \vec{e_z} \tan(\theta)\right). \tag{2.45}$$

This result is interesting, as one can observe that the horizontal component of the force does not depend on the angle θ. Therefore, whether the plate is inclined or not, the horizontal component of the pressure force will only depend on the water depth d.

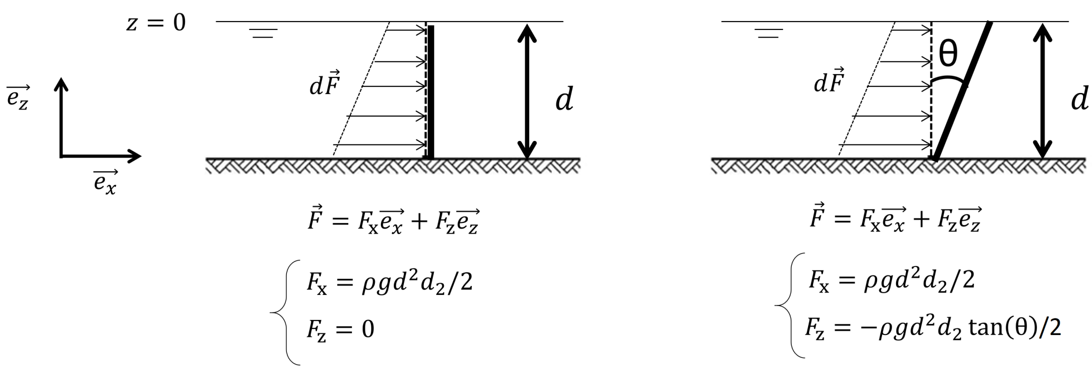


Figure 2.14: *Whatever the angle θ, the horizontal component of the pressure force F_x is always the same.*

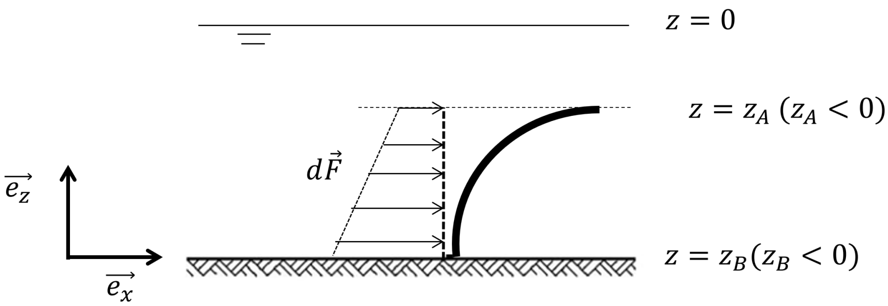
The vertical component of the force will depend on θ: if the plate is placed vertically $\theta = 0$, $\tan(\theta) = 0$, and there will be no vertical component of the force, as expected. If the plate is placed horizontally $\theta = \pi/2$, $\cos(\theta) = 0$, and $\tan(\theta) = \infty$, which means that the solution cannot be applied. In that case, the pressure force is easy to estimate, as the plate is feeling a pressure $\rho g d + P_{\text{atm}}$ over its surface $d_1 \times d_2$.

2.9 Another method to estimate the horizontal pressure force on any surface

Here, we will evaluate the horizontal force on a surface that is not a plate, but a curved surface (of any curvature). We will learn that:

The horizontal component of the pressure force on any surface is the same as the pressure force of the horizontal projection of this surface on a vertical plane.

In the derivation we use the assumption that the fluid density ρ is a constant (the fluid is incompressible). This has as consequence that the pressure and the pressure force on a small section of surface are linear functions of the vertical position z (in Fig. 2.15 you can see that the pressure force on a small section of surface increases linearly with increasing depth).



$$P_A = P(z_A) - P(z = 0) = -\rho g z_A > 0$$

$$P_B = P(z_B) - P(z = 0) = -\rho g z_B > 0$$

Figure 2.15: *Regardless of the shape of the curved surface, the horizontal component of the pressure force F_x is always the same. Note that $P(z = 0) = P_{\text{atm}}$.*

Because it is a linear functions of pressure, the total horizontal force on the curved surface can be evaluated using the average value of the pressure over the corresponding height (see the exercises for a derivation of this relation):

$$F_x = d_2 \int_{z_B}^{z_A} P dz = d_2 \frac{P_A + P_B}{2} \int_{z_B}^{z_A} dz. \tag{2.46}$$

The length d_2 is the length of the curved surface in the direction into the page. Note the pressures P_A and P_B are defined as **gauge pressures** (see exercises), which means they are defined relative to P_{atm}. As we have already discussed, the contribution of the atmospheric pressure cancels out, as it acts everywhere on Earth. We obtain for the horizontal force:

$$F_x = d_2 \frac{P_A + P_B}{2} \left(z_A - z_B\right). \tag{2.47}$$

One can easily verify that this can be further developed into

$$\begin{aligned} F_x &= -\rho g d_2 \frac{(z_A + z_B)}{2} \left(z_A - z_B\right), \\ F_x &= \frac{\rho g d_2}{2} \left(-z_A^2 + z_B^2\right). \end{aligned} \tag{2.48}$$

In the previous section we had $z_A = 0$ and $z_B = -d$.

This fast method can be convenient for quickly evaluating the horizontal force, but the integral method used in the previous exercise is more convenient when, for instance, the reference level $z = 0$ of the z-coordinate is not located at the air/liquid interface as it is here. The integral method is the most general one.

Note that in estuaries or harbours the density ρ is seldom a constant as it depends on the salinity or mud content of the water. In that case, the fast method **cannot** be applied, and the force should be calculated using the integral method with the density $\rho(z)$ being dependent on z.

2.10 Archimedes' principle (1)

In the previous sections, we have been studying the forces due to pressure applied to a surface. We will now study the forces experienced by a volume element inside a fluid. To start, we consider, as we did in the beginning of the Chapter, an element (cube) of fluid inside a larger volume. This time, the cube does not need to be very small. In fact, one can choose to consider a large cube, as shown in Fig. 2.16.

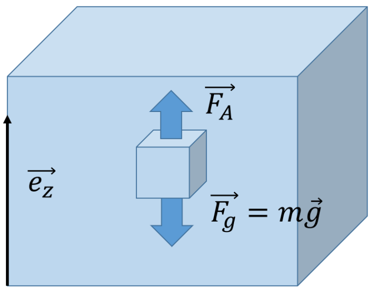


Figure 2.16: *As the cube is at rest, there is a balance of forces, i.e.,* $\vec{F}_A + \vec{F}_g = \vec{0}$.

It is clear that the cube is not moving when the fluid is at rest. As before, we use Newton's second law applied to the fluid cube:

$$\vec{F}_A + m\vec{g} = \vec{0}. \tag{2.49}$$

The force $\vec{F}_A$ is called the **Archimedes' force**. From this relation, it follows that

$$\vec{F}_A = -\rho_{\text{fluid}} V \vec{g} = \rho_{\text{fluid}} V g \vec{e}_z, \tag{2.50}$$

where we have used the fact that the mass of the cube is related to its volume V by $m = \rho_{\text{fluid}} V$ and $\vec{g} = -g\vec{e}_z$.

If we now replace the fluid cube by a cube of same dimensions but made of another material (e.g., wood, iron, clay, plastic), that cube will not be in equilibrium, as its density is different from that of water, as illustrated in Fig. 2.17.

If the cube's density is less than the density of water (wood, plastic), it will start to accelerate upwards, whereas if the cube's density is more than that of water (clay, iron), it will start to accelerate downwards. The total force on the cube is given by

$$\vec{F}_{\text{tot}} = \vec{F}_A + \vec{F}_g = (\rho_{\text{cube}} - \rho_{\text{fluid}}) V \vec{g}. \tag{2.51}$$

One can readily verify that in the case that the cube is made of fluid, we find $\vec{F}_{\text{tot}} = \vec{0}$, as found previously (the cube is at rest). If the density of the cube is less than the density of the fluid,

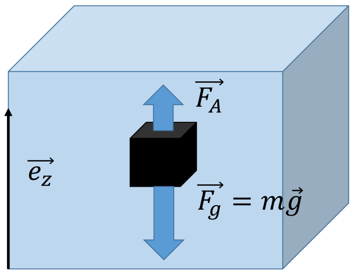


Figure 2.17: *The cube is not in equilibrium and will accelerate (up or down - down in the case of the figure), as there is a disbalance of forces, i.e.,* $\vec{F}_A + \vec{F}_g \neq \vec{0}$.

i.e., $\rho_{\text{cube}} < \rho_{\text{fluid}}$, then the cube will move in the direction opposite to $\vec{g}$; that means it will move in the direction of $\vec{e}_z$. Vice-versa, if the density of the cube is larger than the density of the fluid, i.e., $\rho_{\text{cube}} > \rho_{\text{fluid}}$ then the cube will move in the same direction as $\vec{g}$; that means it will move in the direction opposite to $\vec{e}_z$.

Note that in absence of gravity (as in a space station), there is no Archimedes' force (and no weight): every object (of any density) will therefore remain at rest in the fluid.

2.11 Archimedes' principle (2)

The previous derivation considered the bulk forces applied to a cube element. We will now re-derive Archimedes' force by calculating the surface forces on the cube shown in Fig. 2.18. Let's start with the forces on the four sides of the cube that are perpendicular to the horizontal plane. The force on each side is compensated by an opposite force on the opposite side: the cube will not move in the horizontal plane; if it moves, it will be vertically.

Below the liquid/air interface, the pressure is given by

$$P(z) = \rho_{\text{fluid}} g \,(d - z) + P_{\text{atm}}, \tag{2.52}$$

where d is the height of water between the top of the cube and the liquid/air interface, and z is measured vertically upwards from the top of the cube. The pressure at the top of the cube is $P(z = 0) = \rho_{\text{fluid}} g d + P_{\text{atm}}$. The pressure forces at the top and bottom of the cube can be evaluated as follows:

$$\begin{aligned} \vec{F}_{\text{top}} &= -P(z = 0) a^2 \vec{e_z}, \\ \vec{F}_{\text{bottom}} &= P(z = -a) a^2 \vec{e_z}, \end{aligned} \tag{2.53}$$

where the surface area of a side of the cube is $S = a^2$.

The net force in the vertical direction on the cube is given by

$$\vec{F}_{\text{tot}} = \vec{F}_{\text{top}} + \vec{F}_{\text{bottom}} = -P(z = 0) a^2 \vec{e_z} + P(z = -a) a^2 \vec{e_z}. \tag{2.54}$$

Note the minus sign in front of $\vec{F}_{\text{bottom}}$: the vector $\vec{F}_{\text{bottom}}$ as drawn in Fig. 2.18 is the force that the cube applies to the water underneath. By Newton's third law (for every action, there is an equal and opposite reaction) the force applied by the water on the cube is therefore $-\vec{F}_{\text{bottom}}$. It follows that

$$\vec{F}_{\text{tot}} = -\left[\rho_{\text{fluid}} g d + P_{\text{atm}}\right] a^2 \vec{e_z} + \left[\rho_{\text{fluid}} g \,(d + a) + P_{\text{atm}}\right] a^2 \vec{e_z}, \tag{2.55}$$

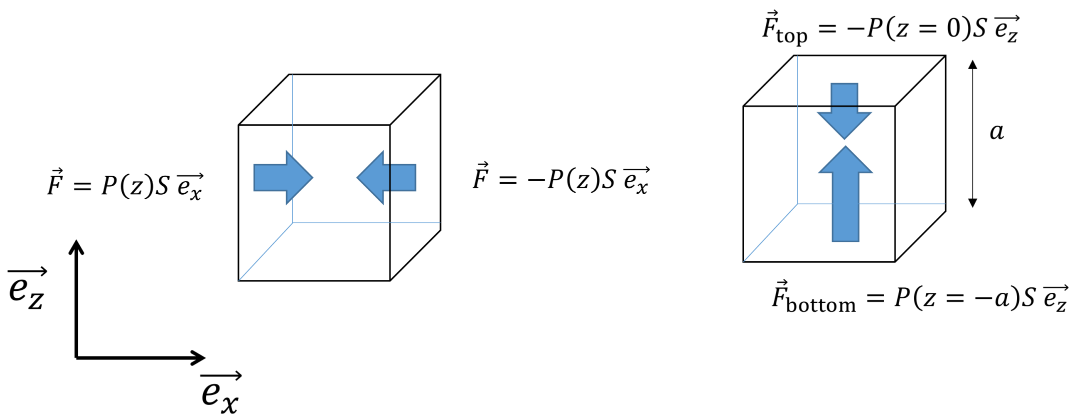


Figure 2.18: *Forces on the sides of a cube in a fluid.*

$$\vec{F}_{\text{tot}} = \rho_{\text{fluid}} g a^3 \vec{e_z}. \tag{2.56}$$

The total force due to the fluid on the cube is Archimedes' force, and, as expected, is equal to

$$\vec{F}_{\text{tot}} = \rho_{\text{fluid}} g V \vec{e_z}, \tag{2.57}$$

where $V = a^3$ is the volume of the cube.

From this derivation, it has become clear that **Archimedes' force originates from the fact that the fluid pressure at the bottom of the cube is higher than at the top, as fluid pressure increases with depth**.

2.12 Archimedes' principle (3)

A third derivation of Archimedes' principle can be performed using the general relation derived at the beginning of this Chapter:

$$\frac{dP(z)}{dz} = -\rho_{\text{fluid}} g. \tag{2.58}$$

In this case, we integrate over the side of the cube:

$$\int_{P(z=0)}^{P(z=-a)} dP = -\rho_{\text{fluid}} g \int_{z=0}^{z=-a} dz, \tag{2.59}$$

from which we directly obtain

$$P(z=-a) - P(z=0) = \rho_{\text{fluid}} g a. \tag{2.60}$$

We thus obtain

$$\vec{F}_{\text{tot}} = \left[P(z=-a) - P(z=0)\right] a^2 \vec{e_z}, \tag{2.61}$$

$$\vec{F}_{\text{tot}} = \rho_{\text{fluid}} g a^3 \vec{e_z}, \tag{2.62}$$

In conclusion, there are several ways to derive Archimedes' force – depending on circumstances one of the methods can be more convenient than another.

2.13 Summary

After studying this Chapter you should:

1. Be able to define the pressure force on a surface: $F = PS$, where P is the pressure and S is the surface area.

2. Be able to derive the expression for the pressure force on a volume:

$$d\vec{F} = -\frac{\partial P(x)}{\partial x} V \vec{e_x}$$

   (and similar expressions in other coordinates).

3. Be able to derive the expression

$$\frac{dP(z)}{dz} = -\rho g$$

   and apply it to calculate the pressure in various circumstances.

4. Be able to define the hydraulic head (piezometric head):

$$h = z + \frac{P(z) - P_{\mathrm{atm}}}{\rho g}.$$

5. Understand the principle of communicating vessels and its implications for aquifers.

6. Understand Pascal's law and be able to apply it.

7. Understand Torricelli's experiment.

8. Be able to calculate the pressure forces (horizontal and vertical components) on different structures (planes, inclined planes, curved surfaces).

9. Understand the origin of Archimedes' principle.

10. Calculate the Archimedes' force by integration or geometric arguments.

3 Hydrodynamics

3.1 Introduction

In this Chapter, we are going to review some basic principles related to the dynamics of fluids. When fluids are in motion, in contrast to the previous Chapter (hydrostatics), there is generally a disbalance of forces, leading to a displacement of fluid. When an object is in motion, one can apply Newton's second law to analyse its trajectory. When applying Newton's second law to a fluid, we first have to define an "object" of fluid and the forces that act on it. We briefly discuss the general equations that describe the motion of fluids, the so-called Navier–Stokes equations, the most important set of equations in fluid dynamics. Using the full Navier–Stokes equations is outside the scope of this course. We will simplify the Navier–Stokes equations to derive two important equations used in this course: the Euler and Bernoulli equations. We will also learn that there are two important ways to characterize the movement of a fluid, the Lagrangian approach and the Eulerian approach. Finally, we will derive the principle of conservation of mass.

3.2 The Navier–Stokes equations

Let's consider a small cube of fluid as illustrated in Fig. 3.1.

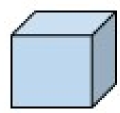


Figure 3.1: *A small cube of fluid of volume* $V = dx \times dy \times dz$.

As in the previous Chapter, this cube is part of a thought experiment: the fluid is not contained in a real cube. Instead, we isolate an element of a fluid in a fictitious cube. In contrast to Chapter 2, the cube is moving, and in this very simple example, we will assume that it is moving vertically only. We do not have to decide if it is moving up or down – we will see later that this depends on the magnitude of the (vertical) pressure force compared to the magnitude of the (vertical) gravity force (equal to the weight of the cube). Variation in the horizontal directions x and y will be ignored altogether.
We can now apply Newton's second law to this (fictitious) cube:

$$m\vec{a} = \vec{F_{\text{tot}}}, \tag{3.1}$$

where m is the mass of the cube, $\vec{a}$ its acceleration and $\vec{F_{\text{tot}}}$ the sum of all the forces acting on the cube. These forces can be classified in two categories: volume forces and surface forces. The volume force is given by the weight:

$$\vec{F_g} = -mg\vec{e_z}. \tag{3.2}$$

Note that we have assumed that $\vec{e_z}$ is pointing upwards (the direction opposite to $\vec{g}$). If you want to adopt an alternative convention, namely $\vec{e_z}$ pointing in the same direction as $\vec{g}$, this will change the results presented in this Chapter, as the minus sign in front of m will have to become a plus sign. It is therefore recommended, in particular when using the Euler and Bernoulli equations we are going to derive in this Chapter, to adopt the convention that the unit vector for the vertical direction (usually the y or z-coordinate in a Cartesian system) is pointing in the direction opposite to $\vec{g}$. The (small) mass m is linked to the density of the fluid by $\rho = m/V$, where V is the (small) volume of the fluid cube. Therefore, we can rewrite this force as

$$\vec{F_g} = -\rho V g\vec{e_z}. \tag{3.3}$$

The pressure force is a surface force, as the pressure acts on the surface of the object, and is given by:

$$\vec{F_p} = -\frac{\partial P}{\partial z}V\vec{e_z}. \tag{3.4}$$

Only the pressure force in the z-direction is considered here. In Chapter 2, we have assumed that $\vec{F_g} + \vec{F_p} = \vec{0}$, i.e., a force balance, leading to $\vec{a} = \vec{0}$, i.e., no acceleration of the fluid. Acceleration and velocity are connected by

$$\vec{a} = \frac{d\vec{v}}{dt}. \tag{3.5}$$

Therefore $\vec{a} = \vec{0}$ implies that $\vec{v}$ is constant in time and, if the fluid is at rest initially, $\vec{v} = \vec{0}$, the fluid will remain at rest.

In this Chapter, we assume that $\vec{F_g} + \vec{F_p} \neq \vec{0}$, i.e., a disbalance of forces. As the fluid is moving, its movement generates friction between different layers (or "cubes") of fluid. This friction force depends on the velocity of the fluid: when the fluid is immobile (at rest), the friction force is zero. The force due to friction (also a surface force) was derived in Chapter 1 and is given by:

$$\vec{F_f} = \eta S\frac{\partial^2 v_z}{\partial z^2}dz\vec{e_z}. \tag{3.6}$$

Here, we have made the assumption that the fluid is moving in the z-direction only and does not vary in the x and y-directions, while the amplitude of the velocity is varying with z and the surface is given by $S = dxdy$. Combining the equations we obtain

$$\rho\vec{a}V = -\rho gV\vec{e_z} - \frac{\partial P}{\partial z}V\vec{e_z} + \eta S\frac{\partial^2 v_z}{\partial z^2}dz\vec{e_z}. \tag{3.7}$$

Realizing that $V = Sdz$, one obtains the one-dimensional Navier–Stokes equation:

$$\rho\vec{a} = -\rho g\vec{e_z} - \frac{\partial P}{\partial z}\vec{e_z} + \eta\frac{\partial^2 v_z}{\partial z^2}\vec{e_z}. \tag{3.8}$$

What does this equation tell us?

1. If v_z is a constant then $\vec{a} = \vec{0}$ (remember we have assumed that the cube is only moving along the vertical axis, and therefore $\vec{v_z} = v_z\vec{e_z}$). In that case

   $$\frac{\partial v_z}{\partial z} = 0, \tag{3.9}$$

   which implies that

   $$\frac{\partial^2 v_z}{\partial z^2} = 0, \tag{3.10}$$

   and we get

   $$\rho g + \frac{\partial P}{\partial z} = 0, \tag{3.11}$$

   which is the equation derived in Chapter 2 for the case of a fluid at rest.

2. If $dP/dz \gg \rho g$, then (3.8) can be rewritten as

$$\rho \vec{a} = -\frac{\partial P}{\partial z}\vec{e_z} + \eta \frac{\partial^2 v_z}{\partial z^2}\vec{e_z}, \tag{3.12}$$

in which case the fluid flow is dominated by the pressure difference that is applied in the vertical direction.

An example is a high-pressure pump that pumps water vertically. Also remember Torricelli's experiment (Chapter 2), which examined the maximum height of water in a closed column. In fact, Torricelli became interested in the experiment after workers at the fountains in Florence discovered that, whatever they tried, they could not pump water above a height of about 10 m. The reason is that the maximum pressure difference a pump using the principle of communicating vessels can deliver is $P_{\text{atm}} - P_{\text{vacuum}} = P_{\text{atm}} - 0$, which corresponds to approximately 10 m of water (or 0.76 m of mercury). Of course, water can be pumped higher than 10 m (otherwise people living in apartments higher than 10 m could not have bathrooms). One simple way to do so would be to have reservoirs every 9 m, and pump water from one reservoir to another, but such a system would not be very convenient. A pump that can pump water to any height is the centrifugal (turbine) pump, which uses the rotation of an impeller to generate a water flow. The rotation of the impeller only depends on the power of the pump and not on gravity.

Equation (3.8) a special case of the **Navier–Stokes equations**. A more general expression in 2D Cartesian coordinates is:

$$\begin{aligned}\rho \vec{a} = -\rho g \vec{e_z} - \left(\frac{\partial P}{\partial x}\vec{e_x} + \frac{\partial P}{\partial y}\vec{e_y} + \frac{\partial P}{\partial z}\vec{e_z}\right) + \eta \left(\frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2}\right)\vec{e_x} \\ +\eta \left(\frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2}\right)\vec{e_y} + \eta \left(\frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2}\right)\vec{e_z}.\end{aligned} \tag{3.13}$$

Remember, the **Navier–Stokes equations** are nothing else than the application of Newton's second law to a volume element of fluid. The forces applied to this element are: 1) gravity, 2) pressure forces and 3) friction forces. The equations were derived by the French engineer and physicist Claude-Louis Navier and Anglo-Irish physicist and mathematician George Gabriel Stokes. The Navier–Stokes equations are useful because they describe the physics of many phenomena of scientific and engineering interest. They may be used to model the weather, ocean currents, water flow in a pipe and air flow around a wing. The Navier–Stokes equations, in their full and simplified forms, help with the design of aircraft and cars, the study of blood flow, the design of power stations, the analysis of pollution, and many other things. We will study simplified formulations of the Navier–Stokes equations, for which analytical solutions exist.
The full Navier–Stokes equations are also of great interest from a purely mathematical perspective. Despite their wide range of practical uses, it has not yet been proven whether solutions always exist in three dimensions and, if they do exist, whether they are smooth, i.e., they are infinitely differentiable at all points in the domain. These are called the Navier–Stokes existence and smoothness problems. The Clay Mathematics Institute has called this one of the seven most important open problems in mathematics and has offered a US$1 million prize for a solution or a counterexample.

In order for the Navier–Stokes equations to be applied, there is still one step to be made: we need to express the acceleration of the fluid as a function of its velocity. Doing so, the Navier–Stokes equations will depend on the fluid velocity (which we are generally interested in for our applications), the pressure applied to the fluid, its viscosity and density, and gravity.

3.2.1 The Euler equations

We are now going to consider a special case of the Navier–Stokes equations, in which we are going to neglect the friction force. As already discussed in Chapter 1, only special fluids have a zero viscosity. For most fluids, one has to consider its viscosity when the fluid is flowing. Nevertheless, in many cases, one can get good approximations of the fluid's bulk behaviour when neglecting friction forces.

An **inviscid flow** is the flow of an **inviscid fluid**, in which the viscosity of the fluid is equal to zero (or assumed to be zero). When viscous forces are neglected, such as in the case of inviscid flow, the Navier–Stokes equations can be simplified to a form known as the Euler equations. These simplified equations are applicable to inviscid flows as well as to flows in which viscosity does not play an important role. Using the Euler equations, many fluid dynamics problems in which viscosity is not important are easily solved. However, the assumption of negligible viscosity is often no longer valid near solid boundaries.

The Euler equations are given by (in 3D):

$$\rho\vec{a} = -\left(\frac{\partial P}{\partial x}\vec{e_x} + \frac{\partial P}{\partial y}\vec{e_y} + \frac{\partial\left(P+\rho g z\right)}{\partial z}\vec{e_z}\right). \tag{3.14}$$

We can make the following observations:

1. When $a_x = 0$, we obtain $\partial P/\partial x = 0$, i.e. no horizontal gradient in pressure. The same holds for the y-direction.

2. When $a_z = 0$, we obtain $\partial P/\partial z = -\rho g$, i.e., there is a constant vertical pressure gradient, due to gravity, known as hydrostatic pressure.

In compact notation, the Euler equation becomes

$$\rho\vec{a} = -\vec{\nabla}\left(P+\rho g z\right), \tag{3.15}$$

where $\vec{\nabla}$ denotes the gradient. This equation is valid for any coordinate system. The piezometric head is defined as (see Chapter 2)

$$h(x,y,z) = z + \frac{P(x,y,z) - P_{\text{atm}}}{\rho g}, \tag{3.16}$$

whereby (realizing that $\vec{\nabla}P_{\text{atm}} = \vec{0}$)

$$\vec{a} = -g\vec{\nabla}h. \tag{3.17}$$

3.2.2 Application of the Euler equations

To illustrate the use of the Euler equations, we consider the following question. What is the pressure P_1 in the situation drawn in Fig. 3.2?

We use the Euler equations is Cartesian coordinates. Only the z-direction needs to be considered in the problem, and therefore:

$$\rho a_z\vec{e_z} = -\frac{d\left(P+\rho g z\right)}{dz}\vec{e_z}. \tag{3.18}$$

The unit vector can be eliminated on both sides of the equation. The acceleration is constant, so it can be taken outside the integral:

$$\rho a_z \int_{z_1}^{z_2} dz = -\int_1^2 d\left(P+\rho g z\right). \tag{3.19}$$

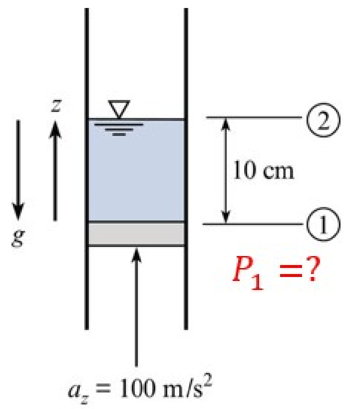


Figure 3.2: *Example of the application of the Euler equations to find pressure.*

The pressure in position 2 is the atmospheric pressure and hence $P_2 = P_{\text{atm}}$. This leads to

$$\rho a_z (z_2 - z_1) = -(P_2 - P_1) - \rho g (z_2 - z_1) . \tag{3.20}$$

Consequently,

$$P_1 = P_{\text{atm}} + \rho (z_2 - z_1)(a_z + g) . \tag{3.21}$$

Using the values given in Fig. 3.2, assuming the fluid is air for example, it is now possible to calculate P_1.

3.3 The relationship between acceleration and velocity

Making use of the concept of a total derivative, the relationship between acceleration and velocity is generally given by

$$\vec{a} = \frac{d\vec{v}}{dt} = \frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \vec{\nabla} \vec{v}. \tag{3.22}$$

This relationship can be found in mathematical textbooks. It expresses the link between a total derivative and partial derivatives. The total derivative $d\vec{v}/dt$ represents the change in velocity of a cube of fluid while it is moving (a so-called **Lagrangian approach**) close to a point M, whereas the partial derivative $\partial\vec{v}/\partial t$ known as the temporal derivative represents the change in velocity of the fluid at a fixed position M (a so-called **Eulerian approach**): it is also called a local derivative for that reason. The term $\vec{v} \cdot \vec{\nabla}\vec{v}$ is the called the advective acceleration ($\vec{a}_{\text{adv}}$) and represents the change in velocity of the fluid between two points M_+ and M_- that are extremely close to M (M_- is just a small distance “before” M and M_+ is small distance “after” M):

$$\vec{a}_{\text{adv}} \simeq \frac{\vec{v}(M_+) - \vec{v}(M_-)}{\Delta t} \simeq \vec{v} \cdot \vec{\nabla}\vec{v}, \tag{3.23}$$

where the approximate identities become identities when we take the limit of the small distances to zero. Note that the advective derivative term is non-linear in $\vec{v}$ (i.e., $\vec{v} \cdot \vec{\nabla}\vec{v}$), which makes it difficult to solve the Navier–Stokes equation analytically in many cases.

In 2D Cartesian coordinates, equation (3.22) becomes:

$$\frac{d\vec{v}}{dt} = \frac{\partial \vec{v}}{\partial t} + \left(v_x \frac{\partial}{\partial x} + v_y \frac{\partial}{\partial y} + v_z \frac{\partial}{\partial z} \right) \vec{v}, \tag{3.24}$$

with $\vec{v} = v_x \vec{e_x} + v_y \vec{e_y} + v_z \vec{e_z}$.

3.4 Lagrangian and Eulerian derivatives

To illustrate the difference between the total (Lagrangian) and partial (Eulerian) derivatives, let's consider a variable that has different values across the space we study. For instance, in a fluid, we could measure the temperature T at any position in time and space.

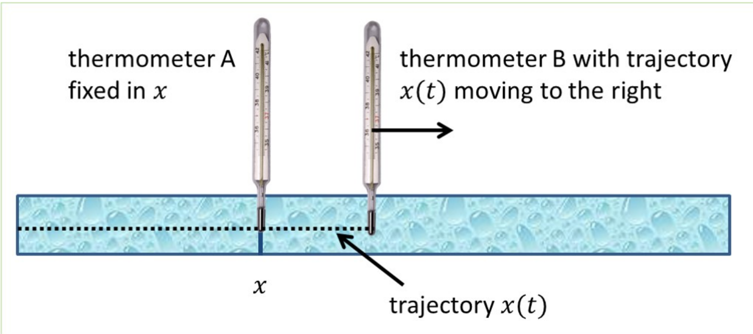


Figure 3.3: *The thermometer A is fixed and represents the Eulerian frame of reference, while the thermometer B is moving and represents the Lagrangian frame of reference.*

For simplicity, we will consider a strip of fluid in the x-direction. We are measuring the temperature with a thermometer A that we place at a fixed position x and a thermometer B that we are moving along x. With thermometer B we get the temperature as a function of $x(t)$, i.e., the trajectory of the thermometer. This implies that for a given time t we are at a specific location x, i.e., x and t are related. In compact notation, we say that we get $T(x(t))$, which corresponds to the temperature along the trajectory $x(t)$. From thermometer A, we obtain the temperature at a fixed position x, where x is now not dependent on time. In this case, in compact notation, we say that $T(x, t)$, which corresponds to the temperature at a fixed position x as a function of time t. Of course, when thermometer B is at the same position as thermometer A, the temperatures are the same. Using the total derivative, we obtain for the rate of change of temperature:

$$\frac{dT}{dt} = \frac{\partial T}{\partial t} + \frac{\partial T}{\partial x}\frac{dx}{dt}, \tag{3.25}$$

where dT/dt is the rate of change in temperature measured with thermometer B, and $\partial T/\partial t$ the rate of change in temperature measured with thermometer A. The term $\partial T/\partial x$ represents the rate of change in temperature with position x (at time t), and $dx/dt = v_B$ is the velocity of thermometer B. From the relationship above, we observe that if the temperature does not depend on time (but is not the same at every position x),

$$\frac{dT}{dt} = \frac{\partial T}{\partial x} v_B, \tag{3.26}$$

which means that even though for a given x the temperature does not change ($\partial T/\partial t = 0$), the thermometer B will indicate a change of temperature in time as it is moving. Only if the thermometer B is not moving or if the temperature is the same at any x, do we also get $dT/dt = 0$.

3.5 Reynolds number and Navier–Stokes

The Reynolds number was already introduced in Chapter 1. We will now connect it to the Navier–Stokes equations. We will use the special case of a 1D Navier–Stokes equations we derived earlier in this Chapter (but the result can be generalized to a more general formulation):

$$\frac{\partial v_z}{\partial t} + v_z \frac{\partial v_z}{\partial z} = -g - \frac{1}{\rho}\frac{\partial P}{\partial z} + \frac{\eta}{\rho}\frac{\partial^2 v_z}{\partial z^2}. \tag{3.27}$$

The Reynolds number Re is the ratio between inertial forces (taken here to correspond to the advective acceleration) and viscous forces. It is given by:

$$\mathrm{Re} = \frac{\left| v_z \frac{\partial v_z}{\partial z} \right|}{\left| \frac{\eta}{\rho}\frac{\partial^2 v_z}{\partial z^2} \right|} == \frac{U\rho L}{\eta} = \frac{UL}{\nu}, \tag{3.28}$$

where the symbol == expresses the fact we perform a dimensional analysis, η is the dynamic viscosity, and ν the kinematic viscosity. The variables U and L denote characteristic velocity and length scales of the fluid. The Reynolds number of an inviscid flow approaches infinity as the viscosity approaches zero. If $\mathrm{Re} \ll 1$, we can omit inertial forces. If, on the other hand, if $\mathrm{Re} \gg 1$, we can omit viscous forces.

3.5.1 Streamlines

Streamlines are families of curves that are very important in fluid dynamics. They represent curves formed by the tangent to the instanteneous velocity at any point in the fluid. Streamlines enable us to study the behaviour of a fluid in the vicinity of objects (cars, bridges, pillars, etc.) and, for instance, to identify regions of space where turbulence plays a role.

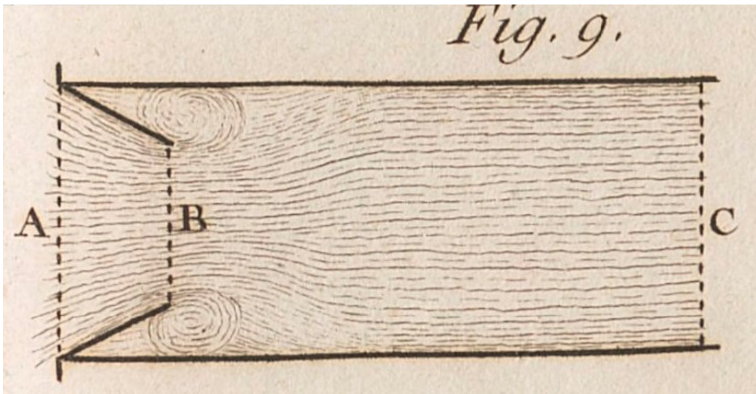


Figure 3.4: *Examples of streamlines drawn by Giovanni Battista Venturi in 1797.*

Streamlines are time independent in the case of a **steady flow**. This means that every particle of fluid (each "cube") will have the same trajectory regardless of time, and hence

$$\frac{\partial v}{\partial t} = 0. \tag{3.29}$$

This equation can be 'translated' to say: at any position in the fluid, the velocity of the fluid does not depend on time. This does *not* mean that $d\vec{v}/dt = 0$ (in fact, $d\vec{v}/dt = \vec{v} \cdot \vec{\nabla}\vec{v}$), as each particle of fluid can have a time-dependent velocity, but all particles passing a given point in the fluid will have the *same* velocity.

In a *steady flow* the Eulerian velocity (so at a fixed position) of the fluid depends on the position but not on time.

The equation we have set up before, i.e.,

$$\frac{d\vec{v}}{dt} = \frac{\partial \vec{v}}{\partial t} \tag{3.30}$$

tells us something else. This equation can be 'translated' to say: at any point in the fluid, the velocity of a particle of fluid crossing that point is equal to the fluid velocity at that point. This implies that the flow only depends on time and not on position ($\vec{a}_{\text{adv}} = \vec{v} \cdot \vec{\nabla}\vec{v} = \vec{0}$). The flow is the same at any point, and when the velocity changes, it changes everywhere in the fluid at the same time. This type of flow is called a uniform flow.

In a *uniform flow* the Eulerian velocity (so at a fixed position) of the fluid can depend on time but not on position.

Steady uniform flow: a flow that does not change with position nor with time.
Example: the flow of water in a pipe of constant diameter at constant flow rate.

Steady non-uniform flow: a flow that changes from point to point in the fluid but does not change in time.
Example: the flow of water in a pipe with decreasing diameter in the direction of the flow.

Unsteady uniform flow: a flow that is the same at any point in the fluid but changes with time.
Example: the flow of water in a pipe of constant diameter connected to a pump which can be used to adjust the flow rate.

Unsteady non-uniform flow: a flow that changes with time and with position.
Example: the flow of water in a pipe with decreasing diameter in the direction of the flow connected to a pump which can be used to adjust the flow rate.

3.6 From Euler to Bernoulli

The Euler equations we have previously considered can be written by replacing the acceleration by the derivative of velocity:

$$\rho \frac{\partial \vec{v}}{\partial t} + \rho \vec{v} \cdot \vec{\nabla}\vec{v} = -\vec{\nabla}\left(P + \rho g z\right). \tag{3.31}$$

We are assuming that the flow is steady, hence

$$\frac{\partial \vec{v}}{\partial t} = \vec{0}. \tag{3.32}$$

This leads to

$$\rho \vec{v} \cdot \vec{\nabla}\vec{v} = -\vec{\nabla}\left(P + \rho g z\right). \tag{3.33}$$

To avoid mathematical complexity, we will now assume that $\vec{v} = v\vec{e_z}$ (the streamlines are along the z-direction, but the result can be generalized for any streamline). The previous equation can be written as

$$\rho v \frac{dv}{dz} = \frac{\rho}{2}\frac{dv^2}{dz} = -\frac{dP}{dz} - \rho g, \tag{3.34}$$

where we have assumed that the fluid has a constant density (the fluid is **incompressible**). This equation can further be developed into

$$\frac{d}{dz}\left(\rho\frac{v^2}{2}+P+\rho gz\right)=0. \tag{3.35}$$

This implies that

$$\rho\frac{v^2}{2}+P+\rho gz = \text{ constant}. \tag{3.36}$$

This last equation is known as **Bernoulli's equation**. This equation is valid **along a streamline in a steady flow without friction and for an incompressible fluid**. Before we use Bernoulli's equation, we must therefore ensure these four assumptions are satisfied (along a streamline, steady and incompressible flow, no friction).

Usually, the Bernoulli's equation is used between two points A and B in a fluid (*along a streamline!*) and can therefore also be written as:

$$\left(\rho\frac{v^2}{2}+P+\rho gz\right)_{\text{at A}}=\left(\rho\frac{v^2}{2}+P+\rho gz\right)_{\text{at B}}. \tag{3.37}$$

The same equation can be written in terms of the piezometric head h:

$$\left(\frac{v^2}{2g}+h\right)_{\text{at A}}=\left(\frac{v^2}{2g}+h\right)_{\text{at B}}. \tag{3.38}$$

Working with piezometric heads h is especially convenient when we are going to study flows in open channels (where the fluid is in contact with atmospheric pressure), as we will see in later Chapters.

3.6.1 Application of Bernoulli's equation

To illustrate the use of Bernoulli's equation, we will consider the following question: when we observe the stream of water coming out of a tap, will the stream get wider or narrower as function of depth?

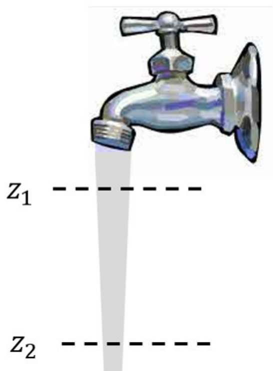


Figure 3.5: *Example of the application of the Bernoulli equation.*

We will apply Bernoulli's equation between the two heights defined in Fig. 3.5:

$$\rho\frac{v_1^2}{2}+P_{\text{atm}}+\rho gz_1=\rho\frac{v_2^2}{2}+P_{\text{atm}}+\rho gz_2. \tag{3.39}$$

Remember that we derived Bernoulli's equation using the convention that the unit vector in the vertical direction (here $\vec{e_z}$) is pointing is the direction opposite to $\vec{g}$, which means that $z_2 < z_1$.

Note that, because the fluid is in contact with air, we use atmospheric pressure for P. As $\rho g z_2 < \rho g z_1$, it follows that $v_1 < v_2$. The flow rate of water (m^3/s) is given by the velocity of water (m/s) multiplied by the surface area S (m^2) of the water perpendicular to the flow direction at that position. As the flow rate of water is constant (we will show why in the next section), it follows that

$$v_1 S_1 = v_2 S_2. \tag{3.40}$$

As $v_1 < v_2$, it follows that $S_1 > S_2$: consequently, the stream of water gets narrower with depth, as in the illustration in Fig. 3.5.

We could also have used the more compact form of Bernoulli's equation expressed in terms of piezometric head:

$$\frac{v_1^2}{2g} + h_1 = \frac{v_2^2}{2g} + h_2. \tag{3.41}$$

In that case,

$$\begin{aligned} h_1 &= z_1 + \frac{P_{\text{atm}} - P_{\text{atm}}}{\rho g} = z_1, \\ h_2 &= z_2 + \frac{P_{\text{atm}} - P_{\text{atm}}}{\rho g} = z_2, \end{aligned} \tag{3.42}$$

and we would have obtained the same result.

3.7 Principle of mass conservation

In physics and chemistry, the law of conservation of mass or the principle of mass conservation states that for any system closed to all transfers of matter (and energy), the mass of the system must remain constant over time. The law of conservation of mass was discovered by the French chemist Lavoisier at the time of the French revolution (where he was himself executed). In 1774, he showed that, although matter can change its state in a chemical reaction, the total mass of matter at the end and at the beginning of every chemical change is the same. Thus, for instance, if a piece of wood is burned to ashes, the total mass remains unchanged if gaseous reactants and products are included. Lavoisier's experiments supported the law of conservation of mass. In France it is taught as Lavoisier's Law and is paraphrased from a statement in his Traité Elémentaire de Chimie:

> *"Nothing is lost, nothing is created, everything is transformed."*

Mikhail Lomonosov (1711–1765) had previously expressed similar ideas in 1748 and proved these in experiments; others whose ideas pre-date the work of Lavoisier include Jean Rey (1583–1645), Joseph Black (1728–1799), and Henry Cavendish (1731–1810).

In fluid mechanics, as taught in this course, conservation of mass only concerns the fact that in a closed system, the fluid cannot loose mass. We have seen in Chapter 1 that a fluid can transfer mass from its liquid form to its vapour form (through evaporation). As stated in Chapter 1, this phase change (or any other) will not be studied in this course, but are of importance when studying for instance the drying of soils or concrete structures.

We have already implicitly used conservation of mass when setting up the Navier–Stokes equations: indeed, we started by defining a fictitious cube of fluid: that cube has a constant mass,

and for this reason we were able to use Newton's second law, from which the Navier–Stokes equations can be derived.
In most exercises using conservation of mass, it is necessary to define a control volume, i.e., a volume in which the change in mass of fluid over time can be calculated. Two types of control volumes exist: the Eulerian control volume, which is a volume fixed in space, and the Lagrangian control volume, which is moving with the fluid (like our little cube). In most exercises the Eulerian control volume will be used.

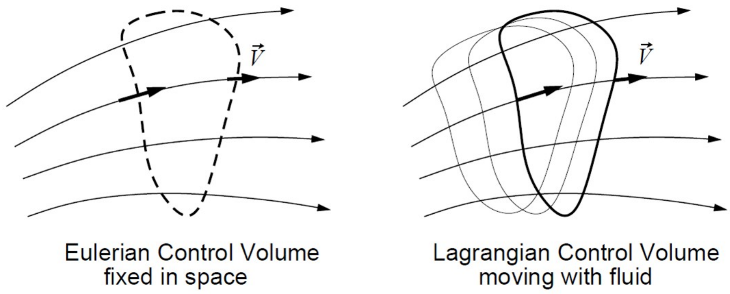


Figure 3.6: *an Eulerian control volume versus a Lagrangian control volume.*

A smart control volume to choose is a volume that corresponds to a tube of streamlines (as in the illustration in Fig. 3.6), as mass then only leaves the control volume in the direction of the streamlines and not in the direction perpendicular to the streamlines.
The law of conservation of mass states that the change in mass in a control volume is equal to the mass entering minus the mass leaving the volume. In mathematical notation, we have:

$$m(t+dt) - m(t) = dm_{\text{in}}(t) - dm_{\text{out}}(t), \tag{3.43}$$

where $m(t+dt) - m(t)$ is the change in mass in the control volume over a small time step dt, $dm_{\text{in}}(t)$ is the small amount of mass entering the control volume, and $dm_{\text{out}}(t)$ the small amount of mass leaving the control volume. This equation can further be developed into

$$\frac{dm}{dt} = \frac{m(t+dt) - m(t)}{dt} = \frac{dm_{\text{in}}}{dt} - \frac{dm_{\text{out}}}{dt}. \tag{3.44}$$

When the mass of fluid in the control volume is not changing, one obtains $dm/dt = 0$, and

$$\frac{dm_{\text{in}}}{dt} - \frac{dm_{\text{out}}}{dt} = 0. \tag{3.45}$$

As the density of the fluid is assumed constant (incompressible fluid), using $\rho = m/V$, where V is the control volume and m the mass of fluid in the control volume, one gets

$$\frac{dV_{\text{in}}}{dt} - \frac{dV_{\text{out}}}{dt} = 0. \tag{3.46}$$

The change in volume over time is called the volume flux or flow rate (m^3/s), which is usually designated by the letter Q. It follows that

$$Q_{\text{in}} = Q_{\text{out}}. \tag{3.47}$$

As shown previously, the volume flux of fluid is equal to the velocity of the fluid at that point multiplied by the surface area perpendicular to that velocity.

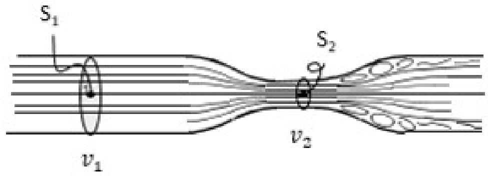


Figure 3.7: *Control volume delimited by streamlines and two cross-sectional surface areas S_1 and S_2.*

In Fig. 3.7 we have chosen the control volume to be the bundle of streamlines with across-streamline boundaries S_1 and S_2. The volume flux of water entering through S_1 is equal to the volume flux of water leaving the volume through S_2:

$$S_1 v_1 = S_2 v_2, \tag{3.48}$$

from which we deduce that $v_1 < v_2$ (as $S_1 > S_2$).

3.8 Application of Bernoulli's equation and mass conservation (1)

We consider the volume flux of water through a pipe of varying diameter, as shown in Fig. 3.8.

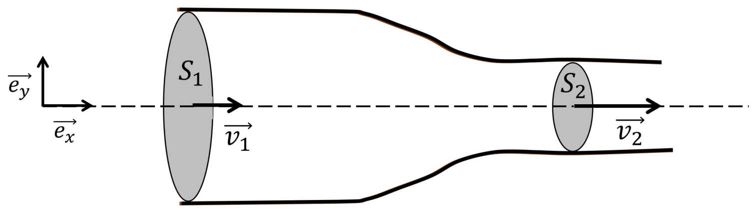


Figure 3.8: *Control volume delimited by a pipe of varying diameter and two surfaces S_1 and S_2, where the pressures are P_1 and P_2.*

In contrast to Fig. 3.7, streamlines are not shown in Fig. 3.8, only the vectors associated with the velocities at locations 1 and 2. Note that the velocity is assumed to be constant over the surfaces S_1 and S_2. This is because we have neglected the forces of friction. If we would have accounted for them (see Chapter 1), the velocity of the fluid would have been zero at the surface of the pipe and at a maximum in the middle of the pipe. We apply the principle of mass conservation, as we assume that the fluid has a constant density (it is incompressible):

$$Q = S_1 v_1 = S_2 v_2. \tag{3.49}$$

We apply Bernoulli along a streamline, assuming a steady flow without friction. The streamline we choose is the dashed straight line along $\vec{e_x}$ (where $z = 0$). This gives:

$$\rho \frac{v_1^2}{2} + P_1 = \rho \frac{v_2^2}{2} + P_2. \tag{3.50}$$

Combining the two equations gives

$$Q^2 = 2 \frac{P_1 - P_2}{\rho} \left(\frac{1}{S_2^2} - \frac{1}{S_1^2} \right)^{-1}. \tag{3.51}$$

Note that the volume flux of water is a consequence of the fact there is a difference in pressure between locations 1 and 2. If $P_1 = P_2$, then $Q = 0$, and the fluid would be at rest.

3.9 Application of Bernoulli's equation and mass conservation (2)

We will now apply the conservation principle to a system that is losing mass, namely, a tank filled with water. The height of water in the tank is decreasing with time. We apply Bernoulli's equation between points A and B in Fig. 3.9 (the streamline is indicated in dashed red):

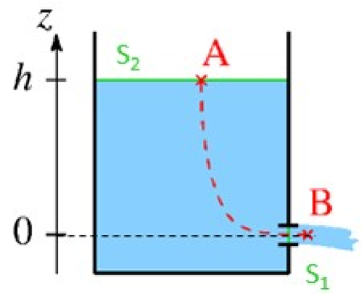


Figure 3.9: *An emptying tank filled with water.*

$$\rho \frac{v_A^2}{2} + P_{\text{atm}} + \rho g z_A = \rho \frac{v_B^2}{2} + P_{\text{atm}} + \rho g z_B. \tag{3.52}$$

From conservation of mass, assuming that the fluid has a constant density, we get

$$S_1 v_B = S_2 v_A, \tag{3.53}$$

and therefore $v_B \gg v_A$ since $S_1 \ll S_2$, from which we deduce that

$$v_B = \sqrt{2g\left(z_A - z_B\right)} = \sqrt{2gh}. \tag{3.54}$$

This velocity is sometimes called **Torricelli's velocity** in honour of the Italian scientist Evangelista Torricelli, who discovered it in 1643 (the same person who studied the maximum height of a fluid in a closed column, see Chapter 2).

3.10 Vena contracta

A **vena contracta** is the point in a fluid stream where the cross-section of the stream is at a minimum and the fluid velocity is at a maximum, such as in the case of a flow leaving a tank through an orifice. In the above example of an emptying tank, this happens after the fluid has left the tank through the bottom orifice. The maximum contraction takes place at a section slightly downstream from the orifice, where the streamlines of the jet are (more or less) horizontal.

A vena contracta is also observed in flow from a tank into a pipe or in the case of a sudden contraction in pipe diameter. The reason for this phenomenon is that fluid streamlines cannot

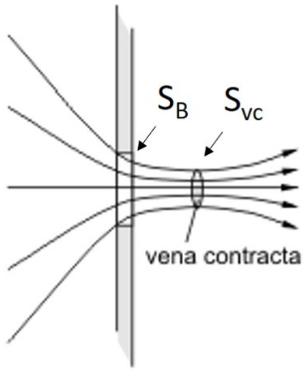


Figure 3.10: *Representation of a vena contracta.*

abruptly change direction. In the case of both the free jet and the sudden change in pipe diameter, the streamlines are unable to closely follow the sharp angle in the pipe/tank wall. The converging streamlines follow a smooth path, which results in the narrowing of the jet (or primary pipe flow).

The diameter of the orifice S_{B} and of the vena contracta S_{vc} are linked by

$$S_{\mathrm{vc}} = \mu S_{\mathrm{B}}, \tag{3.55}$$

where μ is an empirically determined coefficient, which, obviously, should be smaller than 1.

3.11 Summary

After studying this Chapter you should:

1. Understand how to obtain the Navier–Stokes equations from Newton's second law using the friction forces, the weight and the pressure forces on an element of fluid.

2. Know the definition of the Reynolds number and how it relates to the Navier–Stokes equations, i.e.,
$$\mathrm{Re} = \frac{\left| v_z \frac{\partial v_z}{\partial z} \right|}{\left| \frac{\eta}{\rho} \frac{\partial^2 v_z}{\partial z^2} \right|} == \frac{U\rho L}{\eta} = \frac{UL}{\nu}.$$

3. Know the assumption to obtain the Euler equations from the Navier–Stokes equations (i.e., having an inviscid fluid); know how to check that this assumption is satisfied (using the Reynolds number); be able to apply the Euler equations:
$$\rho \vec{a} = -\vec{\nabla} \left(P + \rho g z \right).$$

4. Know and be able to apply (given the flow conditions) the relationship between acceleration and velocity and know the difference between Eulerian (local derivative) and Lagrangian (total derivative) velocities and accelerations.

5. Know the definition of streamlines.

6. Know the differences between steady flow and uniform flow.

7. Know the assumptions required to obtain Bernoulli's equation from the Euler equations (i.e., having a steady flow and considering flow along a streamline); know the 4 assumptions required to use the Bernoulli equation; be able to apply Bernoulli's equation

$$\left(\rho\frac{v^2}{2} + P + \rho g z\right)_{\text{at A}} = \left(\rho\frac{v^2}{2} + P + \rho g z\right)_{\text{at B}}.$$

8. Understand how to obtain the principle of conservation of mass (choose a control volume, draw up a mass balance); apply the fact that the fluid is incompressible (i.e., has a constant density) to obtain between two points:

$$S_1 v_1 = S_2 v_2.$$

9. Use Bernoulli's equation and the mass conservation equation in combination to solve different problems.

10. Know the relationship for the cross-sectional of a vena contracta.

The Dutch corner

Een **uniforme stroming** (of **homogene stroming**) is een stroming die onafhankelijk is van de plaats. Engels: uniform flow.
Een **stationaire stroming** is onafhankelijk van tijd. Engels: steady flow.
Een **eenparige stroming** is zowel onafhankelijk van plaats als tijd. Engels: steady uniform flow.
Een **wrijvingsloze vloeistof** is een vloeistof zonder viscositeit (zonder inwendige wrijving). Engels: inviscid fluid.
Een **ideale vloeistof** is een onsamendrukbaar vloeistof zonder inwendige wrijving. Ideale vloeistoffen ondervinden geen schuifspanningen, viscositeit en warmteoverdracht. Engels: ideal (or perfect) fluid.

4 Pipe flow

4.1 Introduction

In the previous Chapter, the concept of an Eulerian control volume was defined. This is a fixed region of space and time in which the behaviour of the fluid can be studied. The Lagrangian control volume was also introduced, which is moving along with an element of fluid. Both control volumes will be used in this Chapter. In the previous Chapter, we used the Eulerian control volume to apply the principle of mass conservation. In the present Chapter, we are going to estimate the balance of forces on a Eulerian control volume. We will relate the forces to the momentum of the fluid through Newton's second law, which we have already used in Chapter 3 to derive the Euler equations and Bernoulli's equation. We will again use Newton's second law to set up an equation describing conservation of energy. In this case, we will make use of the Lagrangian control volume. From this equation, the concept of power will be introduced, which is important when considering the use of pumps and turbines.

4.2 Forces exerted by fluid flow through a curved pipe

We start by considering the following Eulerian control volume: a section of pipe as represented underneath. The control volume consists of the section of pipe shown in Fig. 4.1 and the fluid contained within; the fluid can leave and enter the control volume through the cross-sectional areas labelled by the blue dashed lines. In mechanics, when the object of study is a solid object with constant mass that is not moving, one can simply use Newton's second law and state that the sum of all external forces on that object is zero. In the example here, fluid is flowing in and out the control volume. It is therefore not generally possible to write, despite the fact that the control volume is not moving, that the sum of all external forces on the control volume should be zero. One should account for the change in momentum related to the in and out-flow of water, known as the change in momentum flux. The reasoning that we are going to develop in the next paragraph stems from the work of Osborne Reynolds (1842-1912). The derivation he presented to express the changes in a control volume is called the Reynolds transport theorem.

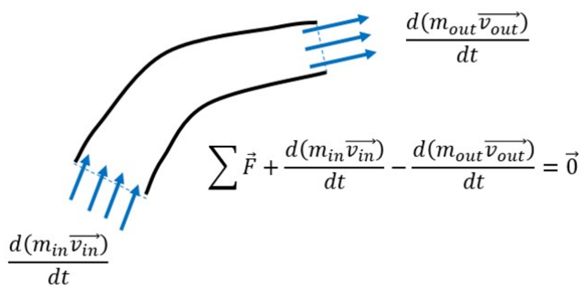


Figure 4.1: *An Eulerian control volume consisting of a section of bent pipe.*

The sum of forces on the control volume is determined as follows. First, we have all the external forces applied to the control volume (the section of pipe and the fluid contained within), which we write as $\sum \vec{F}$. Included in this sum are the force of the fixings that hold the pipe in place, the weight of the pipe, the weight of the fluid in the control volume, and the pressure forces applied by the fluid outside of the control on the fluid inside the control volume at the locations of the blue dashed lines.

According to Newton's second law, the sum of external forces on the control volume $\sum \vec{F}$ must be equal to the rate of change of the momentum (cf. $d(m\vec{v})/dt$) contained inside the control volume, which in turn must be equal to the difference in momentum fluxes entering and leaving the control volume. The **momentum flux** is the rate at which momentum is transported by the fluid. Equivalently, the **volume flux** is the rate at which volume is transported by the fluid. The momentum flux entering the control volume through the left boundary is given by:

$$\vec{F}_{\text{in}} = \frac{d\left(m_{\text{in}}\vec{v}_{\text{in}}\right)}{dt}, \tag{4.1}$$

where we have used the symbol of a force $\vec{F}_{\text{in}}$, because the quantity has dimensions of a force. However, $\vec{F}_{\text{in}}$ is not a force, it is a momentum flux! Similarly, the momentum flux leaving the control volume is given by:

$$\vec{F}_{\text{out}} = \frac{d\left(m_{\text{out}}\vec{v}_{\text{out}}\right)}{dt}. \tag{4.2}$$

Newton's second law now requires:

$$\sum \vec{F} = \frac{d\left(m_{\text{out}}\vec{v}_{\text{out}}\right)}{dt} - \frac{d\left(m_{\text{in}}\vec{v}_{\text{in}}\right)}{dt}. \tag{4.3}$$

Note the sign: if the momentum flux leaving the control volume is greater than the momentum flux entering (i.e., $d(m_{\text{out}}\vec{v}_{\text{out}})/dt - d(m_{\text{in}}\vec{v}_{\text{in}})/dt > 0$), a positive net force (i.e., $\sum \vec{F} > 0$) must have been applied to create this increase in momentum.

In the case of a steady flow, in which the velocities are constant in time, we have

$$\begin{aligned}\frac{d\left(m_{\text{in}}\vec{v}_{\text{in}}\right)}{dt} &= \frac{dm_{\text{in}}}{dt}\vec{v}_{\text{in}},\\ \frac{d\left(m_{\text{out}}\vec{v}_{\text{out}}\right)}{dt} &= \frac{dm_{\text{out}}}{dt}\vec{v}_{\text{out}}.\end{aligned} \tag{4.4}$$

As mass in the control volume is conserved (the pipe is always completely filled) and the fluid is incompressible, we know that the mass fluxes entering and leaving the control volume must be equal:

$$\frac{dm_{\text{in}}}{dt} = \frac{dm_{\text{out}}}{dt}. \tag{4.5}$$

We then obtain

$$\sum \vec{F} = \frac{dm_{\text{out}}}{dt}\left(\vec{v}_{\text{out}} - \vec{v}_{\text{in}}\right). \tag{4.6}$$

This relation is usually called the "momentum balance". We prefer to call it a "force balance". One should simply recognize (4.6) for what it is: Newton's second law applied to an element of fluid, with therefore both "force" terms (on the left-hand side) and "momentum" terms (on the right-hand side).All the surface forces on the control volume are always directed to the interior of the control volume (as they represent forces applied to the control volume). Only the weight of the control volume is a force directed from the *mass centre* of the control volume to the exterior (vertically downwards).

4.3 Momentum fluxes and pressure forces

Let's consider the momentum fluxes in some more detail. We can express the momentum flux into the control volume $\vec{F}_{\text{in}}$ as a function of the surface area of the entrance of the pipe S_{in} (the blue dashed line on the left in Fig. 4.1). We consider the illustration in Fig. 4.2.

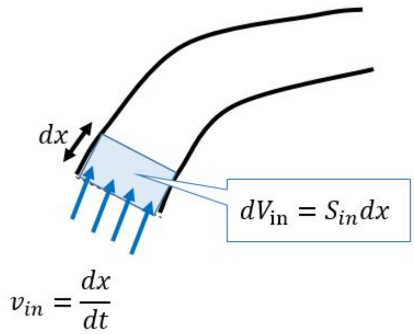


Figure 4.2: *Momentum flux a the entrance of a section of bent pipe.*

We assume, as in the previous section, that the flow is steady, and therefore $\vec{v}_{\text{in}}$ is constant as a function of time. We can relate the mass flux to the volume flux by

$$\frac{dm_{\text{in}}}{dt} = \rho \frac{dV_{\text{in}}}{dt}, \tag{4.7}$$

as we assume a constant density ρ. The small volume dV_{in} represents the amount of water that is entering the pipe during the small time interval dt. During the time dt the water has penetrated into the pipe over a distance dx. This distance dx can be connected to the velocity of the fluid by

$$dx = v_{\text{in}} dt, \tag{4.8}$$

as by definition $v_{\text{in}} = dx/dt$. We now can write

$$dV_{\text{in}} = S_{\text{in}} dx, \tag{4.9}$$

and subsequently

$$F_{\text{in}} = \frac{dm_{\text{in}}}{dt} v_{\text{in}} = \rho \frac{dV_{\text{in}}}{dt} v_{\text{in}} = \rho S_{\text{in}} \frac{dx}{dt} v_{\text{in}}, \tag{4.10}$$

which finally gives

$$F_{\text{in}} = \rho S_{\text{in}} v_{\text{in}}^2. \tag{4.11}$$

This is called the **inward momentum flux**. The same reasoning for F_{out} leads to

$$F_{\text{out}} = \rho S_{\text{out}} v_{\text{out}}^2 \tag{4.12}$$

for the **outward momentum flux**.

In addition to the momentum fluxes, the pressure forces at S_{in} and S_{out} should not be forgotten. The pressure force is given, in the general case, by (see Chapters 1 and 2)

$$F_{\text{pressure}} = PS, \tag{4.13}$$

where P is the pressure that is applied over the surface area S.

It can be convenient to group the momentum fluxes with the external forces applied to the control volume. Remember, the momentum fluxes are not forces but accelerations, in the same way the centrifugal 'force' in curved motion is not a force but an acceleration that requires an actual force, the centripetal force, to be sustained. Combining momentum fluxes and pressures gives for the total 'forces' on S_{in} and S_{out}:

$$F_{\text{in,tot}} = \left(\rho v_{\text{in}}^2 + P_{\text{in}}\right) S_{\text{in}},$$
$$F_{\text{out,tot}} = \left(\rho v_{\text{out}}^2 + P_{\text{out}}\right) S_{\text{out}}. \tag{4.14}$$

This result can be generalized, and the total 'force' on any cross-sectional area of pipe is given by

$$F = \left(\rho v^2 + P\right) S. \tag{4.15}$$

4.3.1 Case where the velocity is not uniform over the surface

In the previous derivation, as indicated in the illustration, we have assumed that v_{in} is the same across the surface area S_{in}. This occurs only for inviscid fluids (fluids with no friction, i.e., no viscosity). In the general case, the velocity v_{in} and the pressure P_{in} would not be constant over S_{in}, as illustrated in Fig. 4.3.

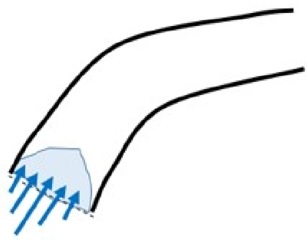


Figure 4.3: *Non-linear and non-uniform velocity profile at the entrance of a bent pipe.*

In that case, the momentum flux and pressure force should be calculated using

$$dF = \left(\rho v^2 + P\right) dS.$$

To set up this equation, we have followed the same line of reasoning as in the previous section. However, we have not considered the whole surface area S but only a small section dS. On the small surface area dS we can assume that v and P are constant. To obtain the whole 'force' F we have to integrate over the whole surface:

$$F = \int \left(\rho v^2 + P\right) dS. \tag{4.16}$$

The same reasoning applies when one wants to calculate the volume flux: S:

$$Q = \int v dS. \tag{4.17}$$

When v is constant over the surface area S, we find the result $Q = vS$, which was derived in the Chapter 3.

Figure 4.4: *Eulerian control volume of a water stream.*

4.4 Force on a control volume in a vertical flow in contact with the atmosphere

We now consider a vertical flow of water as sketched shown in Fig. 4.4. The blue cylinder represents the Eulerian control volume we choose. In a similar way to the one used in the previous section, we can estimate the forces on the control volume. In this case, we are concentrating on the forces on the surfaces at the top and the bottom of the cylinder. On the side of the cylinder the only force acting is the force related to atmospheric pressure, and this cancels out as it is acting everywhere. We assume that the velocity is the same across any horizontal cross-section. We therefore have for the 'force' (actual force combined with momentum flux) on any horizontal cross-section:

$$F(z) = \left(\rho v(z)^2 + P_{\text{atm}}\right) S(z). \tag{4.18}$$

The action of atmospheric pressure also cancels out when we consider the sum of forces on the cylinder (we project the vectors along the vertical axis with the unit vector $\vec{e_z}$ pointing upwards):

$$F_{\text{ext}} - mg - \rho v_{\text{in}}^2 S - P_{\text{atm}} S + \rho v_{\text{out}}^2 S + P_{\text{atm}} S = 0, \tag{4.19}$$

where we include an external force F_{ext}, which represents the force that is required to hold the control volume in place. We obtain

$$F_{\text{ext}} = mg + \rho\left(v_{\text{in}}^2 - v_{\text{out}}^2\right) S, \tag{4.20}$$

with the force $\vec{F}_{\text{ext}}$ pointing upwards. If the water is at rest ($v_{\text{in}} = v_{\text{out}} = 0$), we find that

$$\vec{F}_{\text{ext}} = mg\vec{e_z} \tag{4.21}$$

The external force required for equilibrium is simply the force that oppose the weight of the water in the control volume.

4.4.1 Work done by the forces on an element of fluid in a vertical flow in contact with the atmosphere

Let's look at the previous example in a different way: instead of using a Eulerian control volume (fixed in time and space), we are now using a Lagrangian control volume (moving with the water). This is in fact something we have been doing in Chapter 2 and 3, where the

Lagrangian control volume was simply called "a small cube of fluid" for which we used Newton's second law.

We have to be critical about the word "cube". In the previous example, we used a cylindrical control volume (we could have used any shape). Importantly, when following the cube (or cylinder) as a function of time, its shape will be changing. Why? Because one of the requirements to follow the same portion of fluid as a function of time and be able to use Newton's second law, is to keep its mass (and therefore its volume for constant density) constant. Intuitively, we can already guess what is happening: we know that $v_{\text{in}} < v_{\text{out}}$, so that the bottom of a cylindrical Lagrangian control volume is "falling" faster than its top. To keep the volume of the cylinder constant, it is therefore necessary to decrease the diameter of the cylinder. Note that this is exactly what we found by applying Bernoulli's equation to a streamline in Chapter 3, where we found that $S_{\text{in}} > S_{\text{out}}$. In fact, in the present Chapter, we are slowly building a more general theory than just Bernoulli's equation, and we will show that this general theory reduces to Bernoulli's equation in a special case.

To avoid the problems described above, we will consider a a cylinder with a very small length dz. At time $t = 0$, the cylinder is at position $z = z_A$ and at time $t = t_{\text{end}}$, the cylinder is at position $z = z_B$. The force balance (Newton's second law) on the cylinder is given by:

$$m\frac{d\vec{v}}{dt} = \vec{F}, \tag{4.22}$$

where the small mass of the cylinder is given by $m = \rho S dz$. Note how different this expression is compared to the one we have been using when studying the Eulerian control volume. Here, the change in momentum is associated with the acceleration of a small mass of liquid m with acceleration dv/dt, i.e.,

$$F_{\text{momentum}} = m\frac{dv}{dt}. \tag{4.23}$$

In the case of an Eulerian control volume, we have been transporting mass and momentum through a fixed surface at a given velocity, and the force corresponding to the change in momentum was given by

$$F_{\text{momentum}} = v\frac{dm}{dt}. \tag{4.24}$$

The only external force $\vec{F}$ considered here is the weight of the cylinder $\vec{F} = m\vec{g}$.

As we are following the small cylinder of water as function of time, the cylinder will move from z_A to z_B, so we can calculate the **work done** by the external force (gravity) by taking a dot (or internal product) with the velocity vector:

$$\begin{aligned} m\frac{d\vec{v}}{dt} &= \vec{F}, \\ m\frac{d\vec{v}}{dt}\cdot\vec{v} &= \vec{F}\cdot\vec{v}, \\ \frac{d}{dt}\left(\frac{1}{2}mv^2\right) &= \vec{F}\cdot\vec{v} = \frac{dW}{dt}, \end{aligned} \tag{4.25}$$

where W is the work done by force $\vec{F}$. The velocity of the volume element water (as the water is flowing in the vertical direction) is given by

$$\vec{v} = \frac{dz}{dt}\vec{e_z}. \tag{4.26}$$

Subsequently,

$$\frac{d}{dt}\left(\frac{1}{2}mv^2\right) = F\frac{dz}{dt}, \tag{4.27}$$

as the vectors $\vec{F} = F\vec{e_z}$ and $\vec{e_z}$ are pointing in the same direction. It follows that

$$d\left(\frac{1}{2}mv^2\right) = Fdz = dW. \tag{4.28}$$

This relation can be integrated:

$$W_{A\to B} = \int_{z_A}^{z_B} d\left(\frac{1}{2}mv^2\right) = \int_{z_A}^{z_B} Fdz, \tag{4.29}$$

where $W_{A\to B}$ is the work done by the force $\vec{F}$ between z_A and z_B. Using $F = -mg$ and ignoring work done by pressure, as this is atmospheric everywhere, we obtain

$$W_{A\to B} = \frac{1}{2}m\left(v_B^2 - v_A^2\right) = -\int_{z_A}^{z_B} mgdz. \tag{4.30}$$

Hence

$$\frac{1}{2}m\left(v_B^2 - v_A^2\right) = -mg\left(z_B - z_A\right). \tag{4.31}$$

Rearranging this equation gives

$$\rho\frac{v_B^2}{2} + \rho g z_B = \rho\frac{v_A^2}{2} + \rho g z_A. \tag{4.32}$$

We have therefore re-derived Bernoulli's equation for the case where the pressure is the atmospheric pressure. That seems a lot of work for a result we already know (Bernoulli's equation). Yet, doing this exercise, we gained something. The equation we found, i.e.,

$$d\left(\frac{1}{2}mv^2\right) = Fdz, \tag{4.33}$$

is *much more* general than Bernoulli's equation. To set up this equation (and integrate it), have neither assumed that the flow is steady, nor that there is no friction, nor that the fluid should be incompressible, nor that we should integrate along a streamline. We are now going to start from the general formulation of (4.33) to set up a balance equation that is general (and reduces to Bernoulli under the four assumptions given above).

4.5 Energy balance

The general formulation we start with is (4.25):

$$d\left(\frac{1}{2}mv^2\right) = \vec{F}\cdot dr\vec{e_r}, \tag{4.34}$$

where $\vec{e_r}$ is the unit vector in the direction of the velocity $\vec{v}$, $v = |\vec{v}|$ and dr a small portion of the trajectory of the (Lagrangian) control volume. You have probably already recognized the kinetic energy in this equation:

$$E_k = \frac{1}{2}mv^2. \tag{4.35}$$

The unit of energy is Joule (= J), which is equivalent to the unit of work (= N·m). The force $\vec{F}$ represents all the forces that are operating on the Lagrangian control volume. These forces can be divided into two classes: forces that are conservative and forces that are not. We recall that a force is conservative when the work done by this force depends only on the initial and the final position and not on the trajectory in between. We also recall that the **potential energy** is defined as (minus) the work done by conservative forces. One force that is conservative is weight. As we have just calculated, the work done by weight is equal to $W_{\text{weight}} = -mg\left(z_B - z_A\right)$ when going from z_A to z_B. We had in the previous example

$$\frac{1}{2}m\left(v_B^2 - v_A^2\right) = W_{\text{weight}}. \tag{4.36}$$

We observe that when $z_A > z_B$ then $W_{\text{weight}} > 0$, which implies that $v_B > v_A$. There is a **gain in kinetic energy** (as the velocity of the control volume is increasing). At the same time, there is a **decrease in potential energy**. The potential energy is defined as

$$E_p = mgz. \tag{4.37}$$

We therefore have that

$$\Delta E_p = E_p\left(z_B\right) - E_p\left(z_A\right) = -W_{\text{weight}}. \tag{4.38}$$

We can therefore write, assuming that we have no other force than gravity acting on the control volume, that

$$\begin{aligned} \Delta E_k + \Delta E_p &= 0, \\ \Delta\left(\frac{1}{2}mv^2 + mgz\right) &= 0. \end{aligned} \tag{4.39}$$

This is the same equation we found in the previous section.

In the general case, however, we have another force that is conservative: the force due to pressure. We recall that we have found in Chapter 2 that the pressure force on a small volume is given by

$$\vec{F} = -\frac{\partial P(r)}{\partial r}V\vec{e_r}, \tag{4.40}$$

where V is the volume of the control volume. This leads to

$$W_{\text{pressure}} = -\int_{\vec{r}_A}^{\vec{r}_B} \frac{\partial P(\vec{r})}{\partial r}V\vec{e_r}\cdot dr\vec{e_r}, \tag{4.41}$$

$$W_{\text{pressure}} = -\int_{\vec{r}_A}^{\vec{r}_B} V dP(\vec{r}) = -V\left[P\left(\vec{r}_B\right) - P\left(\vec{r}_A\right)\right]. \tag{4.42}$$

Mass and volume are related by (we assume that the fluid is incompressible, so that density does not change in time): $\rho = m/V$, which implies that

$$\Delta\left(E_k + E_p\right) = W_{\text{except weight and pressure}}/V \tag{4.43}$$

where $W_{\text{except weight and pressure}}$ is the work done by external forces except for weight and pressure, and

$$\begin{aligned} E_k &= \frac{1}{2}\rho v^2, \\ E_p &= \rho g z + P. \end{aligned} \tag{4.44}$$

Note that the dimensions of these "energies" are J/m^3 (corresponding to energy per unit volume); they are energy densities. Work per unit volume $W_{\text{except weight and pressure}}/V$ has the same dimensions (J/m^3).

We now recognize something familiar: the left-hand side of (4.43) is Bernoulli's equation. In the case of Bernoulli's equation, it was assumed that there were no other forces except weight and the pressure force, and hence:

$$\Delta\left(E_k + E_p\right) = 0,$$
$$\Delta\left(\frac{1}{2}\rho v^2 + \rho g z + P\right) = 0, \tag{4.45}$$

Eq. (4.43) can be developed further using the definition of piezometric head:

$$\Delta\left(\frac{1}{2g}v^2 + h\right) = \frac{W_{\text{except weight and pressure}}}{mg}. \tag{4.46}$$

One can verify that indeed $W_{\text{except weight and pressure}}/\left(mg\right)$ has the dimensions of length. We re-write the previous equation as

$$\Delta\left(\frac{1}{2g}v^2 + h\right) = h_+ - h_-, \tag{4.47}$$

where h_+ represents the work done by external forces (expressed as a head difference) that add energy to the system, and h_- the work done by external forces that remove energy from the system.

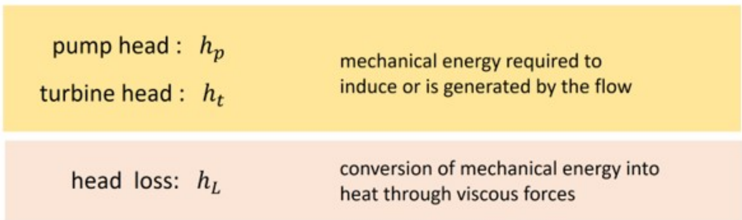


Figure 4.5: Typical sources of head loss or gain in fluid systems.

Energy is removed from the fluid system by the action of a turbine or by the friction force (due to turbulence and viscosity), which dissipates energy into heat:

$$h_- = h_t + h_L. \tag{4.48}$$

Energy is added to the system by the action of a pump:

$$h_+ = h_p. \tag{4.49}$$

4.5.1 Energy balance for realistic flows

The general equation that was set up in the previous section was derived under the assumption that the velocity of the fluid across a section is a constant and therefore equal to its mean value. For real flows, we know (see Chapter 1) that the flow can vary across a section. Therefore, it is convenient for engineering purposes to introduce a (dimensionless)correction factor α, such that

$$\Delta\left(\frac{\alpha}{2g}\overline{v}^2 + h\right) = h_+ - h_-, \tag{4.50}$$

where $\overline{v}$ is the mean value of the velocity of the flow. How can this correction be obtained? By realizing that

$$\alpha = \frac{1}{S}\int \left(\frac{v}{\overline{v}}\right)^3 dS. \tag{4.51}$$

To obtain this, we have followed the same reasoning we have applied above to estimate the volume flux through a section when the velocity v is not uniform across the cross-section. For an inviscid fluid (with zero viscosity), the velocity is constant over a cross-section, and therefore $v = \overline{v}$, leading to $\alpha = 1$. For a Poiseuille flow (this type of flow will be derived in the exercises), the velocity profile is parabolic, and the velocity $v(r)$ as a function of the radial distance r and the mean value $\overline{v}$ of the velocity across any cross-section are given by

$$\begin{aligned} v(r) &= v_{\max}\left(1 - \frac{r^2}{R^2}\right), \\ \overline{v} &= v_{\max}/2, \end{aligned} \tag{4.52}$$

for a pipe of radius R. The correction factor for a circular pipe can therefore be calculated from

$$\alpha = \frac{1}{\pi R^2}\int_0^R \left(\frac{v}{\overline{v}}\right)^3 2\pi r dr, \tag{4.53}$$

which gives $\alpha = 2$ for a Poiseuille flow. For a fully developed turbulent flow in a round pipe the correction factor ranges from 1.04 to 1.11, and it is therefore usually assumed, for simplicity, that $\alpha = 1$. Different types of flows are illustrated in Fig. 4.6.

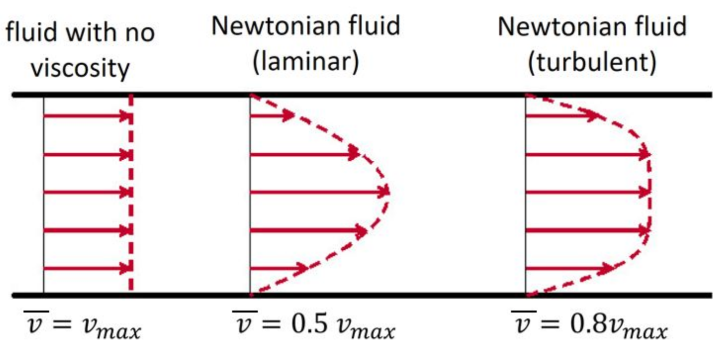


Figure 4.6: *Velocity profiles for different types of flows. For both inviscid fluids* $\alpha = 1$*, for turbulent flows, we have to a good level of approximation that* $\alpha \simeq 1$ *and for laminar flows* $\alpha = 2$.

4.6 Energy gains and losses

Eq. (4.47) is the energy balance of a system that undergoes a transition from a *initial* condition to a *final* condition. One can therefore re-write the equation as follows:

$$H_{\text{initial}} + h_+ = H_{\text{final}} + h_-. \tag{4.54}$$

where $H = v^2/(2g)+h$ is the **total head** also known as the **energy height**. This equation reads as follows: the energy of a system under study examination from H_{initial} to H_{final}, accounting for an input of energy h_+ (delivered for instance by a pump) and a loss of energy h_- (lost for instance by friction or by the action of a turbine). Each gain or loss of head is linked to work done by external forces:

$$h = \frac{W_{\text{initial}\to\text{final}}}{mg}. \tag{4.55}$$

In order to translate the action of a pump or turbine into an energy level or head, one often uses the definition of the **power** of such machine. Power (with units of Watt) is defined as the time derivative of work done

$$\mathcal{P} = \frac{dW}{dt}, \tag{4.56}$$

where we use a calligraphic $\mathcal{P}$ not confuse power $\mathcal{P}$ with pressure P. Using (4.55), we can express the power in terms of a mass flux dm/dt

$$\mathcal{P} = \frac{dW_{\text{begin}\to\text{end}}}{dt} = \frac{dm}{dt}gh. \tag{4.57}$$

The mass flux can be related to the volume flux by:

$$\frac{dm}{dt} = \rho\frac{dV}{dt} = \rho Q, \tag{4.58}$$

where Q (m^3/s) is the volume flux. Finally, we have

$$\mathcal{P} = \rho g h Q, \tag{4.59}$$

which corresponds to the power $\mathcal{P}$ to deliver a head h at a flow rate Q.

The efficiency of pumps and turbines is never 100%, but given by the efficiency factor η (note that unfortunately the symbol for the efficiency factor is the same as the one used for viscosity)

$$\eta = \frac{\mathcal{P}_{\text{output}}}{\mathcal{P}_{\text{input}}}, \tag{4.60}$$

where $\mathcal{P}_{\text{output}}$ is the power output from a machine or system and $\mathcal{P}_{\text{input}}$ is the power input to a machine or system. Note that we always have $\eta \leqslant 1$.

We thus have for a pump:

$$\mathcal{P}_{\text{pump}} = \frac{\rho g h_p Q}{\eta_{\text{pump}}}, \tag{4.61}$$

as the power output is the power resulting from the action of the pump ($\rho g h_p Q$), whereas the power input is the power delivered to the pump from an external source ($\mathcal{P}_{\text{pump}}$).

We have for a turbine:

$$\mathcal{P}_{\text{turbine}} = \eta_{\text{turbine}} \rho g h_t Q, \tag{4.62}$$

as the power output is the power delivered by the turbine ($\mathcal{P}_{\text{turbine}}$), whereas the power input is the energy delivered to the turbine by the system ($\rho g h_t Q$).

For example, consider a system with a pump. An electric power supply provides 750 W to the pump, which delivers only 450 W as output. This means that the efficiency of the pump is

$$\eta_{\text{pump}} = \frac{450}{750} = 0.60 = 60\%, \tag{4.63}$$

and that $\rho g h_p Q = 450$ W. If we know the volume flux of water Q, it is then possible to calculate the energy height or head delivered by the pump to the system h_p.

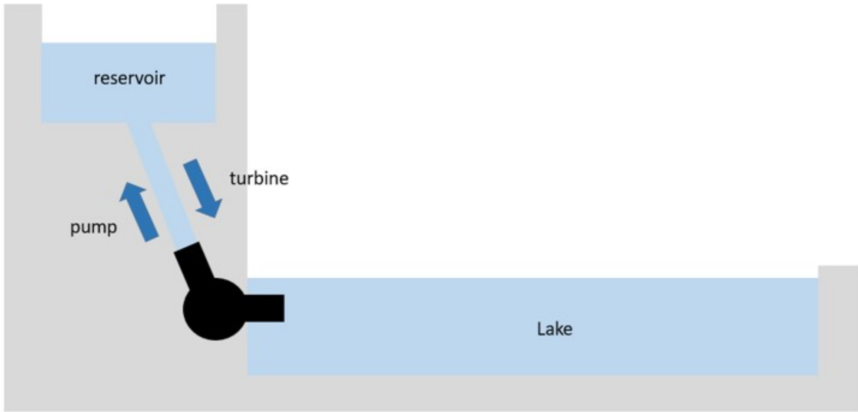


Figure 4.7: *Schematic representation of the action of a pump and a turbine. When the pump mode is activated water is pumped to the reservoir, which adds energy to the system (the reservoir). When the turbine mode is activated, energy is lost from the system.*

Consider instead a system with a wind turbine. Suppose that the wind delivers 1 kW to the turbine, but that the turbine produces only 0.36 kW of electricity. This means that the efficiency of the turbine is

$$\eta_{\text{turbine}} = \frac{0.36}{1} = 0.36 = 36\% \tag{4.64}$$

and that $\rho g h_t Q = 1$ kW. If we know the volume flux of water Q, it is then possible to calculate the energy height or head that is delivered by the system to the turbine h_t.

4.7 The Borda–Carnot equation: sudden enlargement of a pipe

Let's now apply the concepts introduced so far to an important example, which will lead to an equation called the Borda–Carnot equation. This equation describes the loss of mechanical energy of a fluid due to a (sudden) flow expansion (and also due to (sudden) flow contraction, as examined in the next section). The equation is named after the French mathematicians and physicists Jean-Charles de Borda (1733–1799) and Lazare Carnot (1753–1823).

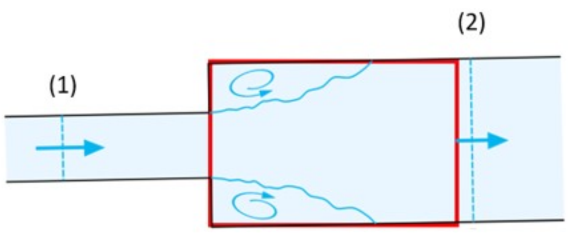


Figure 4.8: *Flow in a pipe with a sudden flow expansion, leading to turbulent losses.*

In the illustration in Fig. 4.8, the Eulerian control volume that we will use to set up the force balance equation is shown in red. When studying the flow of liquids through pipes, there are three equations to consider:

1. the conservation of mass equation,

2. the force balance equation,

3. the energy balance equation,

which we will consider in turn below. It is therefore important to be very familiar with these equations and to know how and when to use them. By "when to use them" we mean under which assumptions they are valid.

4.7.1 Mass balance

We start with conservation of mass. That equation was derived in Chapter 3. We showed that for a fluid of constant density, mass conservation reduces to conservation of volume flux:

$$S_1 v_1 = S_2 v_2, \tag{4.65}$$

where S_1 is the surface area of section 1 and v_1 the velocity through cross-section 1, and the same holds for cross-section 2.

4.7.2 Force balance

The force balance is derived for the Eulerian control volume indicated in red in Fig. 4.8. We recall that

$$\sum \vec{F} = \frac{dm_{\text{in}}}{dt} \left(\vec{v}_{\text{out}} - \vec{v}_{\text{in}} \right). \tag{4.66}$$

Here, we have

$$\frac{dm_{\text{in}}}{dt} = \rho S_1 v_1 = \rho S_2 v_2 = \frac{dm_{\text{out}}}{dt}. \tag{4.67}$$

In the x-direction, we have

$$\sum F_x = \rho S_2 v_2^2 - \rho S_1 v_1^2, \tag{4.68}$$

where $\sum F_x$ includes all the external forces on the control volume along the x-axis. In the vertical direction, we have

$$mg = F_{\text{fixing}}, \tag{4.69}$$

where the weight of the control volume is compensated by fixings to hold the pipe in place. The external forces along the x-axis exerted on the Eulerian control volume are:

$$\sum F_x = P_1 S_2 - P_2 S_2. \tag{4.70}$$

In doing so, we have assumed that the pressure at the beginning of the enlargement is equal to P_1. **This is because the pressure cannot change instantaneously at the location of the sudden enlargement.**

We therefore obtain:

$$\rho S_2 v_2^2 - \rho S_1 v_1^2 = \left(P_1 - P_2 \right) S_2. \tag{4.71}$$

From this equation, using the relation $S_1 v_1 = S_2 v_2$, we can deduce a relation between v_1 and v_2 and eliminate S_2 from the equation:

$$\rho S_2 v_2^2 - \rho S_2 v_2 v_1 = \left(P_1 - P_2 \right) S_2, \tag{4.72}$$

$$\rho v_2 \left(v_2 - v_1 \right) = \left(P_1 - P_2 \right). \tag{4.73}$$

4.7.3 Energy balance

The energy balance equation gives:

$$H_1 = H_2 + h_L, \tag{4.74}$$

where $H_1 = H_{\text{initial}}$, $H_2 = H_{\text{final}}$ and h_L is the energy height or head loss. Furthermore,

$$\begin{aligned} H_1 &= \frac{v_1^2}{2g} + \frac{P_1}{\rho g} + z, \\ H_2 &= \frac{v_2^2}{2g} + \frac{P_2}{\rho g} + z. \end{aligned} \tag{4.75}$$

As the tube is horizontal, we have used $z = z_1 = z_2$. We obtain after simplification

$$\frac{v_1^2}{2g} + \frac{P_1}{\rho g} = \frac{v_2^2}{2g} + \frac{P_2}{\rho g} + h_L. \tag{4.76}$$

Using the relation between v_1 and v_2 obtained from the force balance (4.73), we get

$$\frac{v_1^2}{2g} + \frac{1}{g} v_2 (v_2 - v_1) = \frac{v_2^2}{2g} + h_L, \tag{4.77}$$

which gives

$$v_1^2 + v_2^2 - 2 v_1 v_2 = 2 g h_L, \tag{4.78}$$

from which we obtain the **Carnot equation** for head loss

$$h_L = \frac{(v_1 - v_2)^2}{2g}. \tag{4.79}$$

More generally, head losses are written in the form

$$h_L = k_L \frac{v^2}{2g}, \tag{4.80}$$

where k_L is a constant that is dependent on the type of head loss. The velocity v can be either v_1 or v_2, which depends on the conventions used to determine k_L in practical circumstances, typically from laboratory experiments. Using again the relation $S_1 v_1 = S_2 v_2$, we can rewrite h_L as

$$h_L = \left(1 - \frac{S_1}{S_2}\right)^2 \frac{v_1^2}{2g} = \left(1 - \frac{S_2}{S_1}\right)^2 \frac{v_2^2}{2g}. \tag{4.81}$$

We therefore find that in the case of an enlargement, using v_1 as the reference velocity,

$$k_L = \left(1 - \frac{S_1}{S_2}\right)^2. \tag{4.82}$$

For the case of a pipe discharging into a large tank, $S_2 \gg S_1$, and we find $k_L = 1$. In other words, for a very large expansion, the head loss equals the velocity head before expansion. This gives:

$$\frac{v_1^2}{2g} + \frac{P_1}{\rho g} = \frac{v_2^2}{2g} + \frac{P_2}{\rho g} + \frac{v_1^2}{2g}, \tag{4.83}$$

$$\frac{P_1 - P_2}{\rho} = \frac{v_2^2}{2}. \tag{4.84}$$

The velocity v_2 in the tank does not depend on the velocity upstream, but only on the difference in pressures between upstream and downstream.

4.8 The Borda–Carnot equation: sudden contraction of a pipe

In the previous section we considered the case of a flow encountering a sudden expansion. When there is a sudden contraction of the pipe, the head loss occurs after the entrance of the smaller portion of the pipe, the *vena contracta*, as shown in Fig. 4.9. At that point, applying the

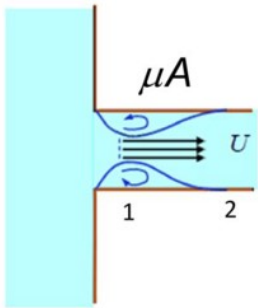


Figure 4.9: Sudden contraction of a pipe.

Carnot principle, the head loss is given by

$$h_L = \left(1 - \frac{A}{\mu A}\right)^2 \frac{U^2}{2g}. \tag{4.85}$$

The velocity U represents the velocity in the pipe, after the contraction. In general, the expression used for a sudden contraction is given by

$$h_L = \left(1 - \frac{1}{\mu}\right)^2 \frac{U^2}{2g}, \tag{4.86}$$

where $\mu = A_{\text{contraction}}/A < 1$ is the ratio between the surface area of the contraction at maximal contraction and the surface area of the pipe at that location (see also Chapter 3, where the vena contracta is defined).

4.9 Example of a sudden contraction (Borda's tube)

In this example we consider a reservoir emptying through a very small tube, known as Borda's tube. We assume the reservoir is very large so that we can neglect the velocity in the reservoir. This example has partially been discussed in Chapter 3, where it was shown that the exit velocity is given by Torricelli's velocity:

$$v_{\text{Torr}} = \sqrt{2gh}, \tag{4.87}$$

where h is the height of water in the reservoir. In the same chapter, the concept of a *vena contracta* was discussed.

We are now going to prove that in the case of a large reservoir emptying through a small tube when head losses can be neglected in the contraction phase (i.e., before the vena contracta), the *vena contracta* coefficient is given by $\mu = S_{CD}/S = 0.5$ with the cross-sectional areas S_{CD}

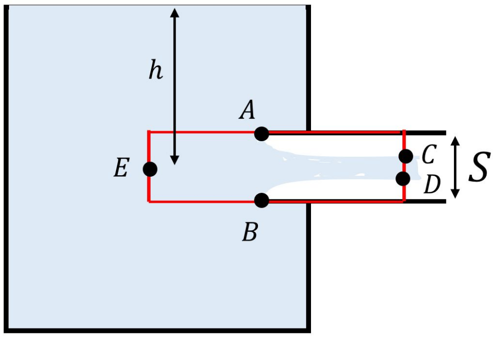


Figure 4.10: Borda's tube.

(vena contracta) and S (small tube) defined in Fig. 4.10. This result was found originally by Borda in 1776.

To demonstrate $\mu = 1/2$ for the idealized case of zero head losses during contraction, we define the Eulerian control volume as given in red in Fig. 4.10. We set up the horizontal force balance for the control volume:

$$(0 + P_E S) - \left(\rho\mu S v_{CD}^2 + P_{\text{atm}} S\right) = 0, \tag{4.88}$$

where $v_{CD} = v_{\text{Torr}}$ is the horizontal velocity at the plane CD. The first term between bracket represents the transfer of momentum at the entrance of the control volume (where we make the assumption that the velocity v_E is much smaller than v_{CD}) and the second term represents the transfer of momentum at the pipe exit. As the water is in contact with the atmosphere, its pressure in the direction perpendicular to the flow is equal to the atmospheric pressure (the small changes in hydrostatic pressures are neglected over the control volume). This equation can be re-written as

$$P_E - P_{\text{atm}} = \rho\mu v_{CD}^2. \tag{4.89}$$

Bernoulli's equation for the system (see also Chapter 3, where the same equation was written in a slightly different form, see (3.52))

$$\frac{0^2}{2g} + \frac{P_E}{\rho g} = \frac{v_{CD}^2}{2g} + \frac{P_{\text{atm}}}{\rho g}. \tag{4.90}$$

It follows from combining (4.89) with (4.90) that

$$\frac{P_E - P_{\text{atm}}}{\rho v_{CD}^2} = \frac{1}{2} = \mu, \tag{4.91}$$

which is what we set out to prove.

Let's now consider Bernoulli's equation between AB and CD, where no energy is lost:

$$\frac{v_{AB}^2}{2g} + \frac{P_{AB}}{\rho g} = \frac{v_{CD}^2}{2g} + \frac{P_{\text{atm}}}{\rho g}. \tag{4.92}$$

In the absence of a *vena contracta*, we have $\mu = 1$. In that case, from mass conservation, assuming that the fluid has a constant density, we get

$$S v_{AB} = S v_{CD}, \tag{4.93}$$

implying that $v_{AB} = v_{CD}$, $P_{AB} = P_{CD} = P_{\text{atm}}$. There is no energy loss, and in fact both kinetic and potential energy remain unchanged between AB and CD. This solution is not very realistic, as it implies that P_{AB} immediately adjusts to the atmospheric pressure, whereas a bit further inside the reservoir, at the same height, we have, because of quasi-hydrostatic conditions, $P_E - P_{\text{atm}} = \rho g h$.

In the case of a *vena contracta* with $\mu = 1/2$, we get that

$$S v_{AB} = S_{CD} v_{CD} = \mu S v_{CD}, \tag{4.94}$$

from which follows that $v_{AB} = v_{CD}/2$. There thus is an increase in kinetic energy between AB and CD, which is compensated by a decrease in potential energy, since by definition there are no energy losses between AB and CD. This decrease in potential energy between AB and CD can be evaluated from

$$\frac{v_{AB}^2}{2g} + \frac{P_{AB}}{\rho g} = \frac{v_{CD}^2}{2g} + \frac{P_{\text{atm}}}{\rho g}, \tag{4.95}$$

$$\frac{P_{AB} - P_{\text{atm}}}{\rho g} = h - h/4 = 3h/4, \tag{4.96}$$

where we have used $v_{CD} = \sqrt{gh}$. This implies that the pressure is decreasing from a value $P_{AB} - P_{\text{atm}} = 3\rho g h/4$ (somewhat smaller than P_E) between AB and CD to reach P_{atm} at exit.

4.10 On the origin of the Borda–Carnot loss

We have derived the general Carnot expression:

$$h_L = \frac{(v_1 - v_2)^2}{2g}. \tag{4.97}$$

This head loss is the consequence of the fact that **pressure cannot change instantaneously at location of the sudden enlargement or contraction**. Indeed, if it could, then the forces along the x-axis exerted on the Eulerian control volume would be (compare this expression with (4.70))

$$\sum F_x = P_2 S_2 - P_2 S_2 = 0, \tag{4.98}$$

which would have as result that

$$v_1 = v_2, \tag{4.99}$$

and therefore

$$h_L = 0. \tag{4.100}$$

The Borda–Carnot losses (due to an incomplete pressure recovery) are linked to **form drag** (also called **pressure drag**), which arises because of the shape of the object. The general size and shape of the body are the most important factors in form drag; bodies with a larger cross-section will have a higher drag than thinner bodies; sleek (“streamlined") objects have lower form drag.

Another type of head loss that will be discussed in the next Chapter is **skin friction** drag. Skin friction drag arises from the friction of the fluid against the “skin” of the object that is moving through it. It thus depends on the viscosity of the fluid. In this Chapter, fluids were considered ideal (and therefore had no viscosity). The sum of form drag and skin friction drag is called parasitic drag, which is the drag that acts on an object moving through a fluid. The other components of total drag, lift-induced drag, wave drag, and ram drag are separate types of drag, and are not components of parasitic drag. They will not be discussed in this course.

4.11 Summary

After studying this chapter you should be able to:

1. Set up the force balance (i.e., "momentum balance") equation, using an Eulerian control volume.
2. Set up the relations for the inward and outward momentum fluxes and the pressure forces to estimate the total "force" on a cross-sectional area of pipe.
3. Understand that the total "force" referred to in point 2 is not a force but the sum of a momentum flux and a force. Be able to explain what the difference between the two is.
4. Apply Newton's second law to set up the force balance for a Lagrangian control volume.
5. Understand the concept of work done by forces and be able to calculate this work.
6. Set up the energy balance for a fluid in motion.
7. Be able to calculate the power input and output of a system and relate it to the efficiency of a related pump or turbine.
8. Be able to setup the Borda–Carnot equation for a sudden enlargement or contraction of a pipe.

The Dutch corner

English	Nederlands
momentum balance	impulsbalans
force balance	krachtenbalans
drag	weerstand
form drag	vormweerstand
skin friction	huidweerstand

5 Open-channel flow

5.1 Introduction

In this Chapter, we are going to use the theoretical knowledge we have acquired so far to solve specific examples encountered in civil engineering applications, open-channel flows. In the previous Chapters, we have been concentrating on flows in pipes, in which the fluid fills the whole section of the pipe. Pipe flows are very important for many applications in industry, but civil engineers very often have to work with open-channel flows, which are flows that are in contact with air and have a surface. We will learn how flow conditions can influence water depth, as illustrated in Fig. 5.1.

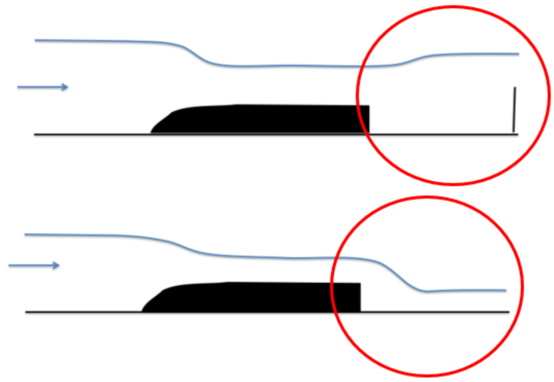


Figure 5.1: *Example of different open-channel flows over an obstacle: despite the fact that the volume flux coming from the left is the same in both cases, two types of flow are possible after the obstacle. We are going to study this in this Chapter and in more detail in Chapter 7.*

5.2 Energy balance

We start by recalling the **energy height** definition (see Chapter 4):

$$H = \left(\frac{v^2}{2g} + z + \frac{P}{\rho g}\right) = \frac{v^2}{2g} + h, \tag{5.1}$$

and apply it to the open-channel system represented in Fig. 5.2. We will consider a channel of constant width b and assume that the height of the bed is constant in the across-channel direction (i.e., in the direction into the page). In doing so, we can also assume that the flow only depends on the horizontal position x. The height of the water is defined as h_0, and in Fig. 5.2 does not depend on x, which is not realistic when the water is flowing, and the bed is not horizontal. This will be discussed further in this Chapter (see final subsections) and in more detail in Chapter 7.
The piezometric head is given by

$$h = z + \frac{P - P_{\mathrm{atm}}}{\rho g} = h_0 = h_b(x) + \mathrm{d}(x), \tag{5.2}$$

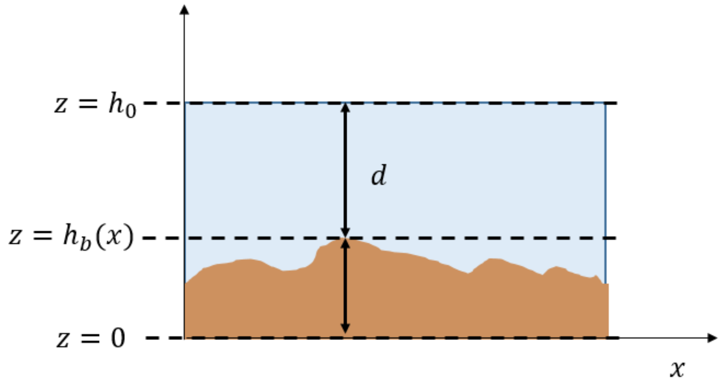


Figure 5.2: *Definitions of the different heights in an open channel.*

where $h_b(x)$ is the height of the bed, and $\mathrm{d}(x)$ is the water depth. The velocity v (m/s) is linked to the volume flux or flow rate Q (m^3/s) by

$$v = \frac{Q}{\mathrm{d}b}, \tag{5.3}$$

where b is the width of the channel (in the direction into the page). Consequently,

$$H = \frac{(Q/b)^2}{2g\mathrm{d}^2} + h_b + \mathrm{d}. \tag{5.4}$$

The **specific energy height** E is defined by $H = E + h_b$, where H is the **energy height**, so that

$$E = \frac{(Q/b)^2}{2g\mathrm{d}^2} + \mathrm{d}. \tag{5.5}$$

The reason we define E here (and did not need it in the previous Chapter) is that, as shown in Fig. 5.2, h_b (the position of the channel bed relative to the $z = 0$ reference level) is very difficult to know in practice. As we can guess, what is important is the distance $\mathrm{d} = h_0 - h_b$, as intuitively we know that the flow of water will get larger when d is reduced (as volume is conserved). Usually, one defines the flow rate q per unit width:

$$q = Q/b, \tag{5.6}$$

and therefore

$$E = \frac{q^2}{2g\mathrm{d}^2} + \mathrm{d}. \tag{5.7}$$

This equation can also be rewritten as

$$q = \sqrt{2g\left(E\mathrm{d}^2 - \mathrm{d}^3\right)}. \tag{5.8}$$

5.3 Different types of flows

5.3.1 Froude Number

We are now going to discuss one of the most important dimensionless numbers in hydrodynamics and see how to apply it in various examples. We will recall from Chapter 4 the two contributions (momentum flux and pressure) to the equation for conservation of momentum:

$$F = \int \Big(\rho v^2 + P\Big)\, dS, \tag{5.9}$$

where the integral is performed over the cross-sectional area. The term ρv^2 is the momentum flux, but can also be associated with the kinetic energy of the system. The pressure P represents the pressure over the cross-section, which is a function of depth. We will assume that the only contribution to pressure is due to gravity and that thus $P = \rho g(h_0 - z)$, i.e., the pressure is hydrostatic. Therefore, we can estimate the 'force' F, assuming for simplicity that v is constant over the whole cross-section:

$$F = \int_{h_b}^{h_0} \Big(\rho v^2 + P\Big)\, b dz, \tag{5.10}$$

$$F = \int_{h_b}^{h_0} \Big(\rho v^2 + \rho g\,(h_0 - z)\Big)\, b dz, \tag{5.11}$$

$$F = \rho b \mathrm{d} v^2 + \rho g b \frac{\mathrm{d}^2}{2}. \tag{5.12}$$

The 'force' per unit of width is thus given by

$$F/b = \rho q^2/\mathrm{d} + \frac{1}{2}\rho g \mathrm{d}^2. \tag{5.13}$$

There are two terms in the expression of the 'force': the first term represents the contribution of the momentum flux or kinetic energy and the second term corresponds to the contribution of pressure or the potential energy. The ratio between the two is the square of the **Froude number:**

$$\mathrm{Fr} = \frac{v}{\sqrt{g\mathrm{d}}}. \tag{5.14}$$

This number also measures the ratio between the velocity of the fluid v and the velocity of small long waves propagating on the surface of the fluid (with shallow-water phase speed $c = \sqrt{g\mathrm{d}}$).

In 1828, the French engineer Bélanger introduced the ratio of the flow velocity to the square root of the gravity acceleration times the flow depth, i.e., the Froude number. When this ratio was less than unity, the flow behaved in a tranquil or 'fluvial' way. When the ratio was greater than unity, Bélanger observed a 'torrential' flow. Subsequently, in 1852, the French naval architect Frederic Reech quantified the resistance of floating objects when towed at a given speed and showed the Froude number to be of importance, as it determined the behaviour of waves along the ship's hull. The English engineer William Froude (1810-1879) was unaware of this work and redid Reech's work, which resulted in the fact that the ratio $v/\sqrt{g\mathrm{d}}$ was credited to William Froude. The French, of course, call this number the Reech–Froude number (in this course the number will be called the Froude number).

The optimum of the 'force' F with respect to d is obtained by:

$$\frac{d\,(F/\mathrm{d})}{d\mathrm{d}} = \rho g \mathrm{d}\left(1 - \frac{q^2}{g\mathrm{d}^3}\right) = \rho g \mathrm{d}\left(1 - \mathrm{Fr}^2\right), \tag{5.15}$$

$$\frac{d\,(F/b)}{d\mathrm{d}} = 0 \rightarrow \mathrm{Fr} = 1. \tag{5.16}$$

When $\mathrm{Fr} = 1$, we have $v = \sqrt{g\mathrm{d}}$. The depth at which this optimum occurs is called the **critical depth** and is given by

$$\mathrm{d}_g = \left(q^2/g\right)^{1/3}. \tag{5.17}$$

Similarly, the optimum specific energy height E with respect to d is obtained by:

$$\frac{d\,(E)}{d\mathrm{d}} = -\frac{q^2}{2gd^3} + 1 = 1 - \mathrm{Fr}^2 = 0, \tag{5.18}$$

and occurs when $\mathrm{Fr} = 1$. Inverting (5.8), we can evaluate the volume flux of water q as function of the depth d for a constant energy E, as shown in Fig. 5.3.

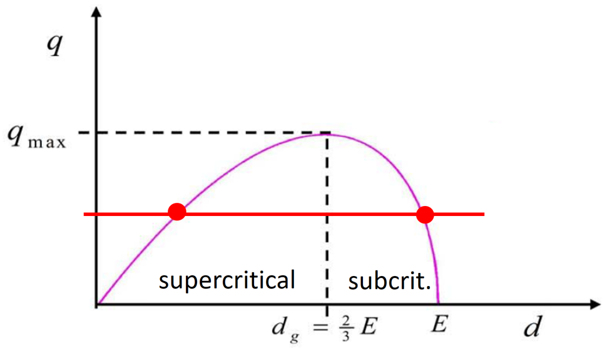


Figure 5.3: *Volume flux of water as function of depth for a given energy E. Note that for a given volume flux, there are two flows possible (red bullets): the flow can either be super-critical ('torrential') or sub-critical ('fluvial').*

The maximum volume flux (from setting $dq/d\mathrm{d} = 0$, keeping E constant) occurs when $\mathrm{d} = \mathrm{d}_g = 2E/3$:

$$q_{\max} = \frac{2}{3}E\sqrt{g\frac{2}{3}E}. \tag{5.19}$$

For a given energy, there are two possible volume fluxes of water: how do we know which one will occur? We will discuss the two solutions in the section below. When we plot the energy E as a function of depth d for a constant flux q, we obtain the same two solutions, as shown in Fig. 5.4.

5.4 Super-critical, critical and sub-critical flows

The Froude number enables us to characterize three types of flows: super-critical ($\mathrm{Fr} > 1$), critical ($\mathrm{Fr} = 1$) and sub-critical ($\mathrm{Fr} < 1$) flows. As was observed by Bélanger when $\mathrm{Fr} < 1$, the flow behaves in a 'fluvial' or tranquil way, and when $\mathrm{Fr} > 1$ it behaves like a torrent (a rapid flow). The four different types of flow are illustrated in Fig. 5.5 (the flow is from left to right).

Sub-critical flow, fluvial motion ($\mathrm{Fr} < 1$): the pressure or potential-energy term is dominant and

E	F/b	v	d
$\to \mathrm{d}$	$\to g\rho\mathrm{d}^2/2$	$< \sqrt{g\mathrm{d}}$	$> \mathrm{d}_g$

Critical flow ($\mathrm{Fr} = 1$): No term is dominant and

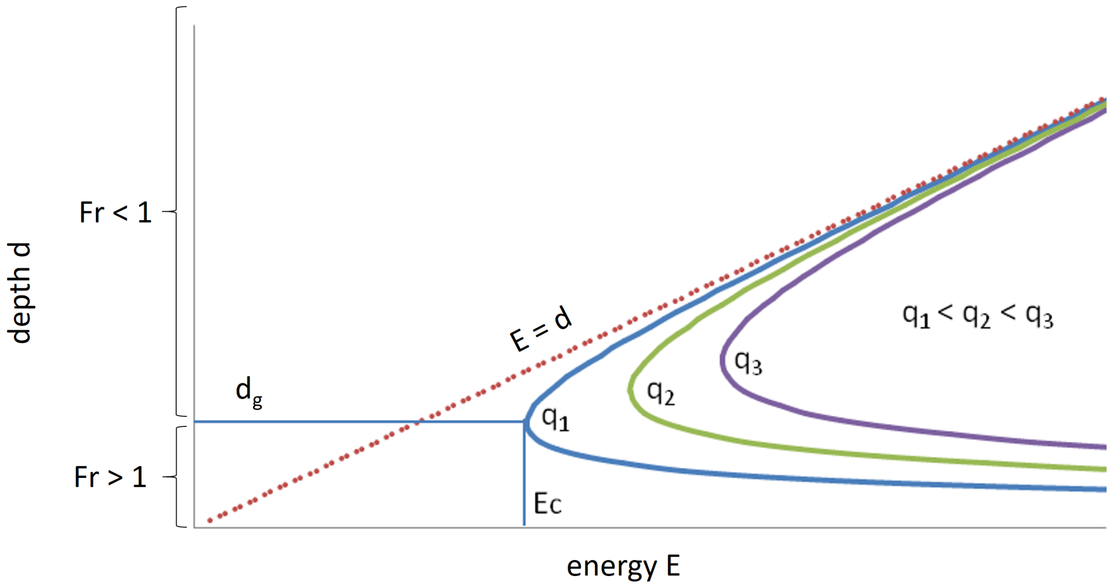


Figure 5.4: *Depth of water as a function of energy for different volume fluxes q. Note that for a given energy, two depths are possible: one corresponds to a super-critical (torrential flow, Fr > 1), the other to a sub-critical flow (fluvial flow, Fr < 1).*

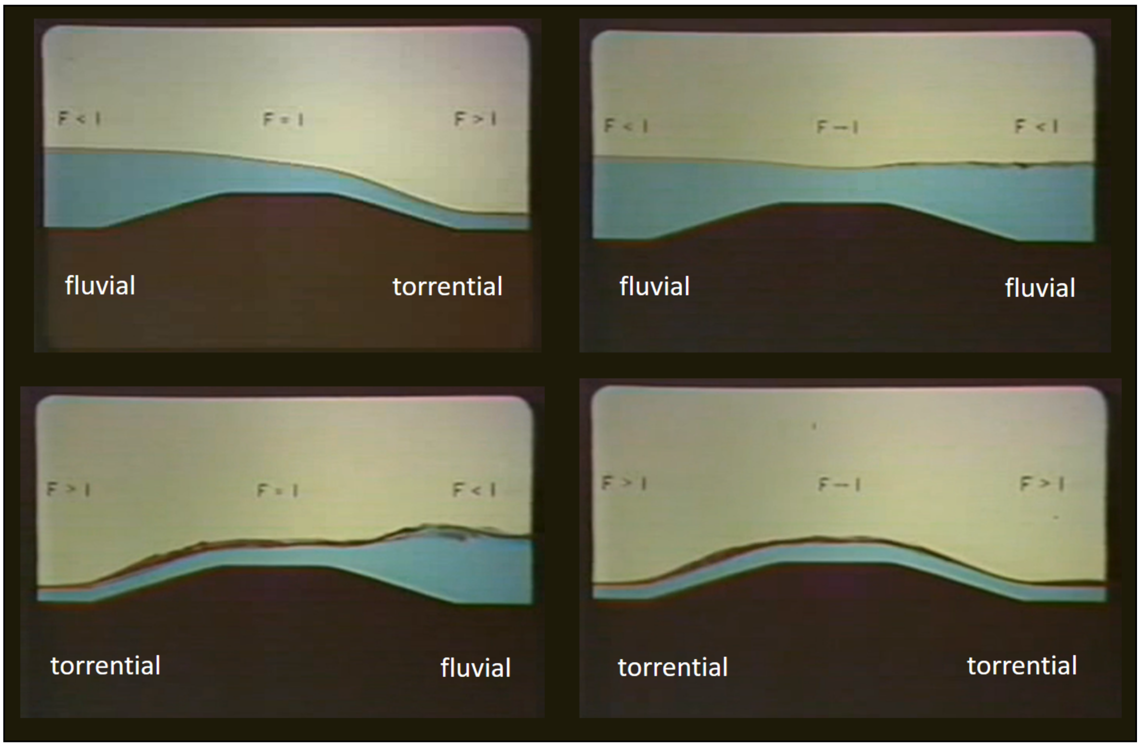


Figure 5.5: *Examples of flows over a weir (a weir is a small dam built across a river to control the upstream water level). The fluvial motion is associated with a* ***tranquil flow*** *and the torrential motion with a* ***rapid flow****.*

E	F/b	v	d
$= 3\mathrm{d}_g/2$	$= 3g\rho\mathrm{d}^2/2$	$= \sqrt{g\mathrm{d}}$	$= \mathrm{d}_g$

Super-critical flow, torrential motion (Fr $>$ 1): the momentum flux or kinetic-energy term is dominant and

E	F/b	v	d
$\to q^2/(2gd^2)$	$\to \rho q^2/\mathrm{d}$	$> \sqrt{g\mathrm{d}}$	$< \mathrm{d}_g$

Let's now examine some examples in which we analyse the behaviour of water over different hydraulic structures.

5.4.1 Spillway (1)

A spillway is a structure used to provide the controlled release of water from a dam or levee downstream, typically into the riverbed of the dammed river itself. In the United Kingdom, spillways may be known as overflow channels. Spillways ensure that water does not damage parts of the structure not designed to convey water.
We have seen that for the same volume flux, two energy levels are possible. This is represented in Fig. 5.6, where, depending on the Froude number we have case (I) or case (II).

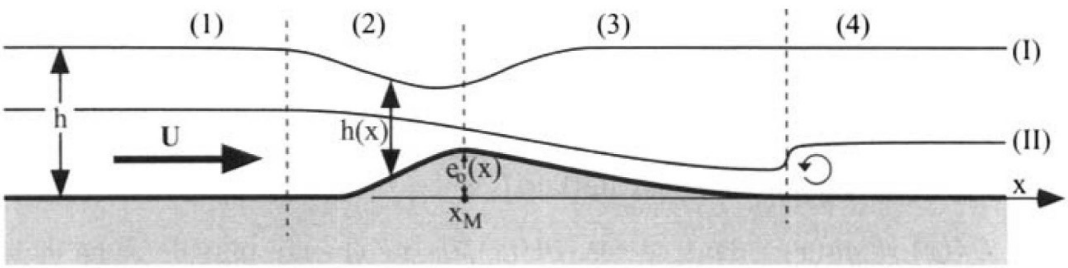


Figure 5.6: *Examples of two possible types of flows over a spillway: case (I) and case (II). Note that the minimum of h(x) for case (I) should be at x_M; this is not accurately represented in the figure.*

We apply Bernoulli along the streamline that goes along the surface of the water from a position far upstream to the position x:

$$P_{\text{atm}} + \frac{1}{2}\rho U^2 + \rho g h = P_{\text{atm}} + \frac{1}{2}\rho U(x)^2 + \rho g \left(h\left(x\right) + e_0(x)\right). \tag{5.20}$$

Conservation of mass (volume) gives

$$Uh = U(x)h(x). \tag{5.21}$$

Differentiating both equations with respect to x, we obtain

$$\rho U(x)\frac{\partial U}{\partial x} + \rho g\frac{\partial h}{\partial x} + \rho g\frac{\partial e_0}{\partial x} = 0, \tag{5.22}$$

$$U(x)\frac{\partial h}{\partial x} + h(x)\frac{\partial U}{\partial x} = 0. \tag{5.23}$$

Combining these equations, we obtain

$$\frac{1}{U(x)}\frac{\partial U}{\partial x}\left(-gh(x) + U^2(x)\right) + g\frac{\partial e_0}{\partial x} = 0. \tag{5.24}$$

We assume that the velocity in part (1) of the flow (as indicated in Fig. 5.6) is slow enough and h high enough, so that

$$-gh + U^2 < 0, \tag{5.25}$$

which implies that Fr < 1 (we have a fluvial motion). In part (2) of the flow, before point x_M, $\partial e_0/\partial x > 0$, $\partial U/\partial x > 0$ and $-gh(x) + U^2(x) < 0$. When the flow of water arrives at the position x_M, the height of the bottom is highest, and $\partial e_0/\partial x = 0$. The equation reduces to

$$\frac{1}{U(x)}\frac{\partial U}{\partial x}\left(-gh(x) + U^2(x)\right) = 0. \tag{5.26}$$

This equation has two solutions: case (I) and case (II).

Case (I) In case (I) the flow stays sub-critical, as the spillway is not high enough for the flow to become critical. At $x = x_M$, we have $\partial U/\partial x = 0$ and, using the derivative of the mass conservation equation,

$$\frac{\partial h}{\partial x} = 0. \tag{5.27}$$

At the point x_M the height of water h is smallest, the velocity U highest, but the Froude number remains lower than 1 (Fr < 1). After x_M, in part (3) of the flow, the height h increases again and U reduces again to the value it had in part (1) of the flow. For case (I), we have Fr < 1 in parts (3) and (4) of the flow. Case (I) corresponds to the 'transition' from fluvial to fluvial.

Case (II) In case (II), the flow becomes critical, and $U^2(x_M) = gh(x_M)$; in that case, Fr $=$ 1 at x_M. The term $\partial U/\partial x$ does not change sign, and therefore U continues increasing (and consequently h decreasing) after point x_M. In part (3) of the flow, we get $-gh(x) + U^2(x) > 0$ and $\partial e_0/\partial x < 0$. We have Fr > 1 in part (3). What happens in part (4) will be discussed in a following section. Case (II) corresponds to the transition from fluvial to torrential.

5.4.2 Spillway (2)

The example corresponding to case (I) studied in the previous section is illustrated in Fig. 5.7 in terms of energy levels. We will relate the energy levels at different positions to the volume flux of water and the heads.

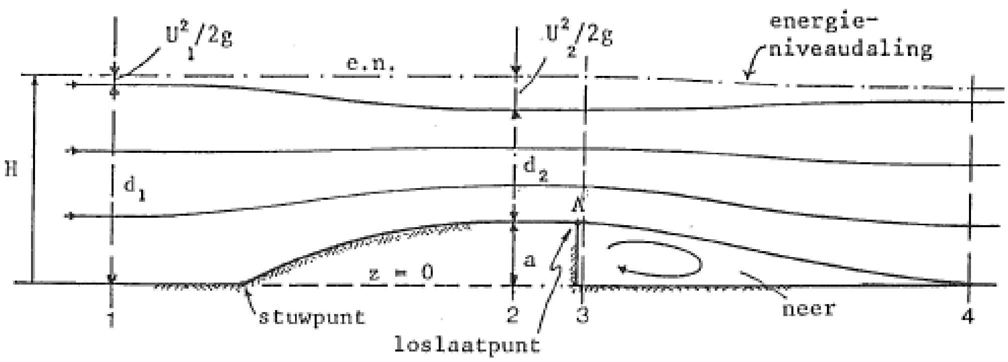


Figure 5.7: *Flow over a spillway from an energy perspective.*

We assume that the flow is 2D as before. To study this type of problem, we are going to select an Eulerian control volume: the volume of water in between the cross-sections d_1 and d_2. In that control volume, there is an acceleration of the fluid as

$$v_1 = \frac{\mathrm{d}_2}{\mathrm{d}_1} v_2 < v_2. \tag{5.28}$$

To set up this equation, we have used mass conservation (see Chapter 4) and the fact that

$$\begin{aligned} S_1 &= \mathrm{d}_1 b, \\ S_2 &= \mathrm{d}_2 b. \end{aligned} \tag{5.29}$$

We are neglecting the energy losses in the control volume, which implies that

$$H_1 = \left(\frac{1}{2g}\alpha\overline{v_1}^2 + h_1\right) = H_2 = \left(\frac{1}{2g}\alpha\overline{v_2}^2 + h_2\right). \tag{5.30}$$

As the flow is turbulent, we will take $\alpha = 1$. The piezometric heads are given by

$$\begin{aligned} h_1 &= \mathrm{d}_1, \\ h_2 &= \mathrm{d}_2 + a. \end{aligned} \tag{5.31}$$

We recall that by definition (see Chapter 2):

$$h_1 = \frac{P_1 - P_{\text{atm}}}{\rho g} + z_1 = \frac{P_{\text{atm}} - P_{\text{atm}}}{\rho g} + \mathrm{d}_1 = \mathrm{d}_1, \tag{5.32}$$

and h_2 can be obtained in a similar fashion.
The volume flux is given by

$$q = \frac{Q}{b} = v_1 \mathrm{d}_1 = v_2 \mathrm{d}_2. \tag{5.33}$$

We can estimate the specific energy

$$\begin{aligned} E_2 &= H_2 - a, \\ E_1 &= H_1, \end{aligned} \tag{5.34}$$

and therefore

$$\begin{aligned} v_2 &= \sqrt{2g\left(E_2 - \mathrm{d}_2\right)}, \\ q &= \sqrt{2g\left(E_2 \mathrm{d}_2^2 - \mathrm{d}_2^3\right)}. \end{aligned} \tag{5.35}$$

5.4.3 Hydraulic jump

In the example Spillway (1), there was a transition between parts (3) and (4) that we are going to discuss now. This flow transition is called an **hydraulic jump**.
An hydraulic jump occurs when the fluid is decelerating: its kinetic energy is transformed into potential energy and turbulence, and the water level increases abruptly: this happens for example when you open the tap, as shown in Fig. 5.8.
Several scenarios are possible when you open a tap: if the velocity of the fluid is less than its critical velocity, no hydraulic jump is possible. If the velocity of the fluid is slightly larger than the critical velocity, the transition takes the form of waves. When the velocity of the fluid is increased further, the transition becomes more and more abrupt, until the transition

Figure 5.8: *Example of a stationary hydraulic jump (transition from rapid flow to tranquil flow).*

Figure 5.9: *A tidal bore is an hydraulic jump that occurs when the incoming tide forms a wave (or waves) of water that travel up a river or narrow bay against the direction of the current.*

zone breaks down and winds around itself. When this happens, an hydraulic jump appears, in conjunction with turbulence and the formation of waves and roll waves.

In the case of a kitchen sink, the hydraulic jump is stationary as in Fig. 5.8, but the hydraulic jump can also move, as in the case of a tidal bore, as shown in Fig. 5.9.

Let's now analyse, using the conservation laws we have been studied, what happens at an hydraulic jump. The important parameters for the hydraulic jump are defined in Fig. 5.10.

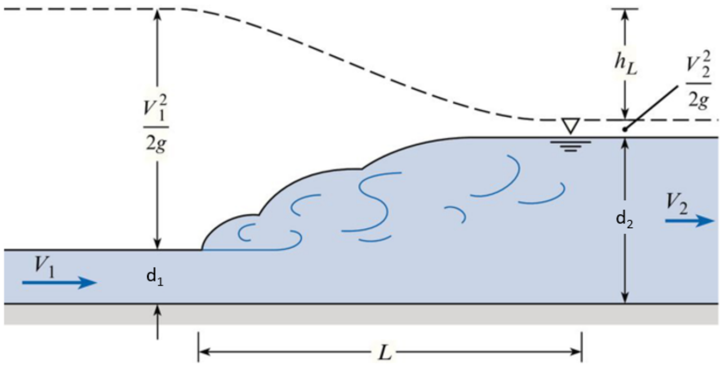


Figure 5.10: *An hydraulic jump.*

Applying conservation of volume:

$$q = v_1 \mathrm{d}_1 = v_2 \mathrm{d}_2. \tag{5.36}$$

Applying conservation of energy:

$$H_1 = H_2 + h_L, \tag{5.37}$$

with

$$\begin{aligned} H_1 &= \frac{1}{2g} v_1^2 + d_1, \\ H_2 &= \frac{1}{2g} v_2^2 + d_2, \end{aligned} \tag{5.38}$$

as we have realized that

$$\begin{aligned} h_1 &= d_1, \\ h_2 &= d_2. \end{aligned} \tag{5.39}$$

Choosing as Eulerian control volume the portion of fluid between locations 1 and 2, we obtain for the balance of forces:

$$\left(\frac{1}{2}\rho g b \mathrm{d}_1^2 + \rho b \mathrm{d}_1 v_1^2\right) - \left(\frac{1}{2}\rho g b \mathrm{d}_2^2 + \rho b \mathrm{d}_2 v_2^2\right) = 0. \tag{5.40}$$

Using conservation of volume together with this last equation, we get

$$\left(\frac{1}{2}\rho g \mathrm{d}_1^2 + \rho q^2/\mathrm{d}_1\right) - \left(\frac{1}{2}\rho g \mathrm{d}_2^2 + \rho q^2/\mathrm{d}_2\right) = 0, \tag{5.41}$$

from which we deduce that, using the definition of the critical depth,

$$\mathrm{d}_g^3 = \frac{q^2}{g} = \mathrm{d}_1 \mathrm{d}_2 \frac{\mathrm{d}_1 + \mathrm{d}_2}{2}, \tag{5.42}$$

from which we can further show that

$$\frac{\mathrm{d}_2}{\mathrm{d}_1} = \frac{1}{2}\left[\sqrt{1+8\mathrm{Fr}_1^2} - 1\right], \tag{5.43}$$

where we have used

$$\mathrm{Fr}_1 = \frac{v_1}{\sqrt{g\mathrm{d}_1}}. \tag{5.44}$$

This behaviour of $\mathrm{d}_1/\mathrm{d}_2$ as function of Fr_1^2 corresponds well to the measurements shown in Fig. 5.11.

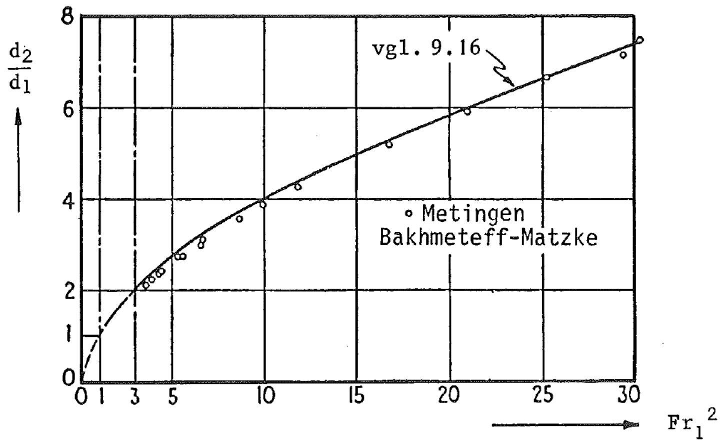


Figure 5.11: *The ratio of water heights before and after the hydraulic jump as a function of the Froude number of the flow before the jump.*

The velocities v_1 and v_1 can also be expressed as functions of the heights:

$$v_1 = \sqrt{g\frac{\mathrm{d}_1+\mathrm{d}_2}{2}\frac{\mathrm{d}_2}{\mathrm{d}_1}},$$
$$v_2 = \sqrt{g\frac{\mathrm{d}_1+\mathrm{d}_2}{2}\frac{\mathrm{d}_1}{\mathrm{d}_2}}. \tag{5.45}$$

5.5 Summary

After studying this chapter you should be able to:

1. Understand the concept of specific energy height and set up the relevant equation:

$$E = \frac{q^2}{2g\mathrm{d}^2} + \mathrm{d}.$$

2. Be able to define the Froude number:

$$\mathrm{Fr} = \frac{v}{\sqrt{g\mathrm{d}}}$$

   and understand what this number represents.

3. Understand the concept of critical depth and how it is related to the Froude number.
4. Understand what super-critical ("torrential") and sub-critical ("fluvial") flows are.
5. Be able to set up relations between volume fluxes, energy levels and water heights for spillways and hydraulic jumps.

The Dutch corner

English	Nederlands
spillway	overlaat
submerged flow	onvolkomen overlaat gestuwde afvoer verdronken afvoer
free flow	volkomen overlaat ongestuwde afvoer
weir	stuw
dike	dijk
crest	kruin
slope	talud
bank	oever
hydraulics	waterloop
bed forms	beddingvormen
flume	meetgoot
gate	schuif
stream	stroom

6 Friction

6.1 Introduction

In Chapter 4, we have discussed how to set up balance equations when systems experience energy losses and gains. Gain of energy occur when mechanical energy is 'injected' into the system (for instance by a pump). Loss of energy occurs when we let the fluid do work and convert energy to perform a useful task (for instance in turbines and windmills). We studied what happens in particular in the case of a sudden enlargement or contraction of a pipe. We have also mentioned that energy is lost due to friction forces and we have mentioned turbulence and viscosity. In this Chapter, we are going to study the action of friction in more detail. First, we will study the action of friction on the flow itself, in the case of flows through pipes. Then we are going to study how to account for forces of friction on an object located in a flow.

6.2 Laminar and turbulent flows in pipes

Laminar flows are especially important for studying flows in small conduits or in porous media (such as concrete, sandstone or clayey material). We have seen in the exercises that this type of flow is dominated by viscous forces. Laminar flow is a flow in which the fluid motion occurs in layers. In Chapter 1, we have connected the transition between laminar and turbulent flows to the Reynolds number. Reynolds discovered, by studying flows in pipes, that the flow undergoes the transition from laminar to turbulent when Re is of the order of 2000. Reynolds' experiment is illustrated in Fig. 6.1.

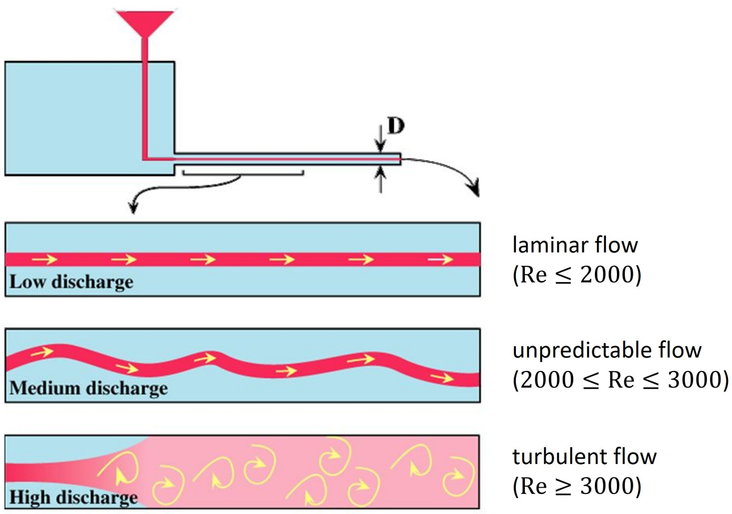


Figure 6.1: *Reynolds's experiment: depending on the velocity of the flow, the regime can be either laminar or turbulent. There is also a Reynolds number range for which the flow is unpredictable: it can change back and forth between laminar and turbulent.*

The transition from laminar to turbulent depends on four main parameters: the velocity of the fluid v, a characteristic length scale (here, the diameter of the pipe D), the density ρ and the viscosity η of the fluid. These are indeed the parameters appearing in the definition of the Reynolds number:

$$\mathrm{Re} = \frac{v\rho D}{\eta}. \tag{6.1}$$

6.2.1 Developing and fully developed flows

In Chapter 4, we have studied the losses due to a sudden enlargement or contraction of a pipe, which may also occur in a pipe bend: the flow has to adapt to the sudden change. When a flow enters a pipe, it similarly has to adapt to the change before it reaches its fully developed profile, as shown in Fig. 6.2

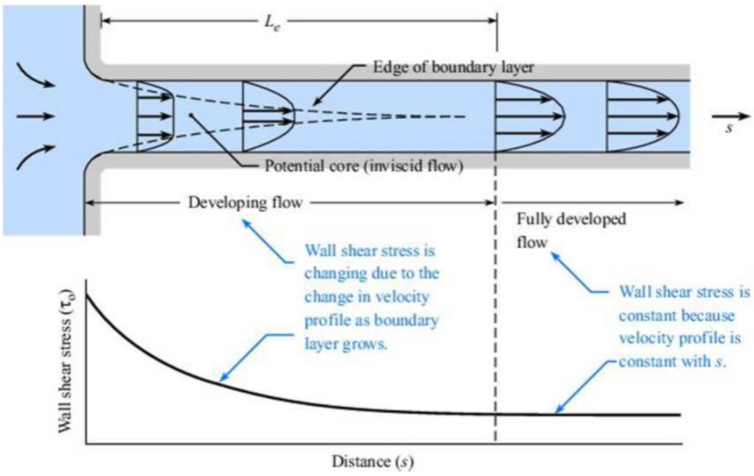


Figure 6.2: *In developing flows, the wall shear stress is changing. It has a constant value when the flow is fully developed.*

The length required to achieve a fully developed flow is given in the Fig. 6.2 by L_e. The link between Re, D and L_e is given by the empirical relationships

$$\begin{aligned} \frac{L_e}{D} &= 0.05\,\mathrm{Re} \quad \text{for laminar flows (Re } \leq 2000\text{)},\\ \frac{L_e}{D} &= 50 \quad \text{for turbulent flows (Re } \geqslant 3000\text{)}. \end{aligned} \tag{6.2}$$

In Fig. 6.2, the profile of the wall shear stress is shown. We refer to Chapter 1 for the definition of shear stress. Because of this shear stress, additional head losses are occuring in the pipe that were not accounted for in Chapter 4. These head losses apply to the fully developed part of the flow and are called **pipe head losses** (sometimes also called major head losses). The losses in the developing part of the flow (due to the passage of liquid through a sudden enlargement or contraction of a pipe) are accounted for in the way studied in Chapter 4. These losses are called **component head losses** or minor head losses.

6.3 The Darcy–Weisbach equation

Pipe head losses can be estimated using the Darcy–Weisbach equation, which we are going to study now. As we have done in Chapter 4, we are going to set up an Eulerian control volume

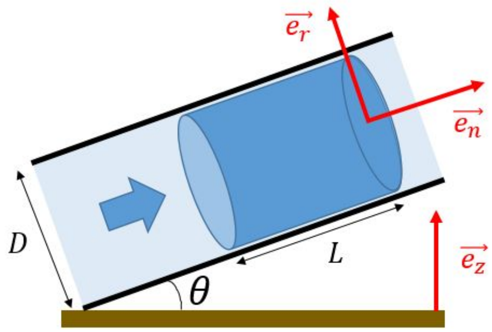


Figure 6.3: *Eulerian control volume in a pipe.*

and study the forces applied to that volume. We recall the general expression we have derived for a steady flow in Chapter 4:

$$\sum \vec{F} = \frac{dm_{\text{in}}}{dt} \left(\vec{v_{\text{out}}} - \vec{v_{\text{in}}} \right). \tag{6.3}$$

We assume that the flow is fully developed and steady. Because of conservation of mass (volume), we have

$$S_{\text{in}} v_{\text{in}} = S_{\text{out}} v_{\text{out}}. \tag{6.4}$$

The surface area of the pipe is constant and $S_{\text{in}} = S_{\text{out}} = S = \pi D^2/4$, from which we deduce that

$$v_{\text{in}} = v_{\text{out}}. \tag{6.5}$$

It follows that (6.3) reduces to

$$\sum \vec{F} = \vec{0}. \tag{6.6}$$

The forces applied to the control volume are (1) the weight of the fluid given by

$$\vec{F_g} = m\vec{g} = -\rho V g \vec{e_z}, \tag{6.7}$$

where $V = SL$ is the volume of fluid, (2) the pressure forces on S_{in} and S_{out}

$$\begin{aligned} \vec{F}_{\text{in}} &= S P_{\text{in}} \vec{e_n}, \\ \vec{F}_{\text{out}} &= -S P_{\text{out}} \vec{e_n}, \end{aligned} \tag{6.8}$$

where we recall from Chapter 4 that the minus sign originates from the fact that we consider the forces on the control volume, and (3) the friction forces due to the shear stresses on the pipe walls, where the liquid experience shearing with the wall. This force is expressed as

$$\vec{F}_{\text{shear}} = -\tau_0 A \vec{e_n}, \tag{6.9}$$

where $A = \pi D L$ is the surface area on which the shear is acting. Note that $\vec{F}_{\text{shear}}$ is directed in the **direction opposite to the flow** (as friction always opposes the flow). Summing up, we obtain

$$-\rho V g \vec{e_z} + S \left(P_{\text{in}} - P_{\text{out}} \right) \vec{e_n} - \tau_0 A \vec{e_n} = \vec{0}. \tag{6.10}$$

Projecting this equation on the axis defined by $\vec{e_n}$, we obtain

$$-\rho V g \sin(\theta) + S \left(P_{\text{in}} - P_{\text{out}} \right) - \tau_0 A = 0. \tag{6.11}$$

By definition

$$\sin(\theta) = \frac{z_{\text{out}} - z_{\text{in}}}{L}, \tag{6.12}$$

which leads to

$$S(P_{\text{in}} + \rho g z_{\text{in}}) - S(P_{\text{out}} + \rho g z_{\text{out}}) - \tau_0 A = 0. \tag{6.13}$$

Using the energy balance equation, we have (recall that $v_{\text{in}} = v_{\text{out}}$)

$$\left(\frac{P_{\text{in}}}{\rho g} + z_{\text{in}}\right) = \left(\frac{P_{\text{out}}}{\rho g} + z_{\text{out}}\right) + h_f, \tag{6.14}$$

where h_f represents the head loss due to friction. Combining (6.13) and (6.14), we obtain

$$h_f = \frac{\tau_0 A}{\rho g S} = \frac{4\tau_0}{\rho g}\frac{L}{D}. \tag{6.15}$$

A dimensionless coefficient, **the friction factor** f is defined by

$$f = \frac{\text{shear stress acting at the wall}}{\text{dynamic pressure}} = \frac{4\tau_0}{\rho v^2/2}, \tag{6.16}$$

from which we deduce the **Darcy–Weisbach equation**

$$h_f = f\frac{L}{D}\frac{v^2}{2g}. \tag{6.17}$$

The Darcy–Weisbach equation is applicable to flows that are fully developed and steady. The equation can be used for either laminar or turbulent flow.

6.4 Laminar flows in pipes

Let's evaluate the velocity profile and friction factor for laminar flows. Instead of considering a tube of length L, we are now considering a small portion of tube of length dl. Using (6.13), we get

$$S(P_l + \rho g z_l) - S(P_{l+dl} + \rho g z_{l+dl}) - \tau_0 \pi D dl = 0. \tag{6.18}$$

The reason we use a small portion of tube dl instead of a long section, is that we can now make use of the mathematical definitions

$$\begin{aligned} \frac{dP}{dl} &= \frac{P_{l+dl} - P_l}{dl}, \\ \frac{dz}{dl} &= \frac{z_{l+dl} - z_l}{dl}, \end{aligned} \tag{6.19}$$

to obtain for the shear stress on the wall of the tube

$$\tau_0 = -\frac{D}{4}\left(\frac{dP}{dl} + \rho g\frac{dz}{dl}\right). \tag{6.20}$$

We can however follow the same reasoning for any tube of radius r and obtain for the shear stress in the fluid at any radius r:

$$\tau(r) = -\frac{r}{2}\frac{d}{dl}(P + \rho g z), \tag{6.21}$$

$$\tau(r) = -\frac{r}{2}\rho g \frac{dh}{dl}, \tag{6.22}$$

where dh is the difference in piezometric head over the portion dl. This equation indicates to us that the shear stress in a tube is zero at its center (where $r = 0$) and increases linearly towards the wall of the cylindrical tube to reach its maximum stress τ_0 for $r = D_0/2$.

We recall the definition of viscosity:

$$\tau(r) = \eta \frac{dv}{dr}, \tag{6.23}$$

which was derived in Chapter 1. It follows from (6.22) and (6.23) that

$$\frac{dv}{dr} = -\frac{r}{2\eta}\rho g \frac{dh}{dl}. \tag{6.24}$$

As dh/dl is constant over any section of the tube, we can integrate this equation to yield

$$v = -\frac{r_0^2 - r^2}{4\eta}\rho g \frac{dh}{dl} + v(r = r_0), \tag{6.25}$$

where $D = 2r_0$. The velocity of the wall is zero, and so is the velocity of the fluid. This is called the **no-slip condition**: $v(r = r_0)$. (Obviously, when there is slip, the velocity of the fluid at the wall will be non-zero, but this will not be studied in this course.) Consequently,

$$v = -\frac{r_0^2 - r^2}{4\eta}\rho g \frac{dh}{dl} \tag{6.26}$$

We refer the exercises, where this parabolic profile is calculated using the Navier–Stokes equations. The velocity is a positive quantity, and one should keep in mind that $dh/dl < 0$. One can show that

$$v_{\max} = v\,(r = 0) = -\frac{r_0^2}{4\eta}\rho g \frac{dh}{dl}, \tag{6.27}$$

and therefore

$$v(r) = v_{\max}\left(1 - \frac{r^2}{r_0^2}\right). \tag{6.28}$$

An illustration of this profile known as **Poiseuille flow** is given in Fig. 6.4

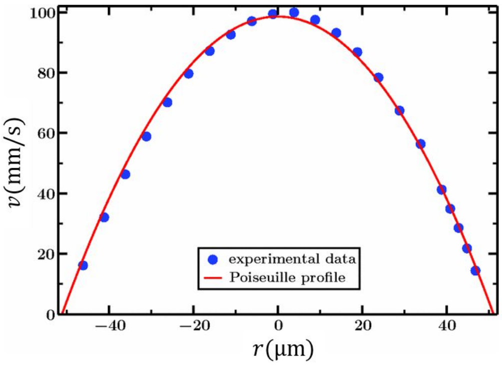


Figure 6.4: *Comparison between a measured and a calculated Poiseuille profile in a capillary tube.*

The mean value of the velocity over the cross-sectional area of the pipe is given by

$$\overline{v} = \frac{v_{\max}}{\pi r_0^2} \int_0^{r_0} \left(1 - \frac{r^2}{r_0^2}\right) 2\pi r dr = \frac{v_{\max}}{2}. \tag{6.29}$$

It follows that

$$\overline{v} = -\frac{r_0^2}{8\eta} \rho g \frac{dh}{dl}. \tag{6.30}$$

To calculate the head loss due to friction, we will make used of (6.14), which for the small tube of length dl can be written as

$$\frac{dh}{dl} = -\frac{h_f}{L}, \tag{6.31}$$

as we assume the head loss to be constant over the length of the pipe. We therefore obtain

$$h_f = \frac{8\eta L \overline{v}}{\rho g r_0^2} = \frac{32\eta L \overline{v}}{\rho g D^2} = f \frac{L}{D} \frac{\overline{v}^2}{2g}. \tag{6.32}$$

The corresponding friction factor is given by

$$f = \frac{64\eta}{\rho D \overline{v}} = \frac{64}{\mathrm{Re}}. \tag{6.33}$$

The friction factor for a laminar flow depends only on the Reynolds number.

6.5 Turbulent flows in pipes

Because of the chaotic motion of fluid particles undergoing turbulent motion, turbulent flows produce high levels of mixing and have a velocity profile that is more uniform (i.e., flatter) than the corresponding laminar velocity profile. Engineers model turbulent flow using an empirical approach, which relies on equations that mathematically capture the behaviour of the flow, but do not rely on first principles. In contrast, we have seen that for laminar flows it was possible to derive equations using first principles (Newton's second law or the Navier–Stokes equations). Empirical solutions have been studied for a long time and can provide a very accurate description of the behaviour of the flow in most cases.

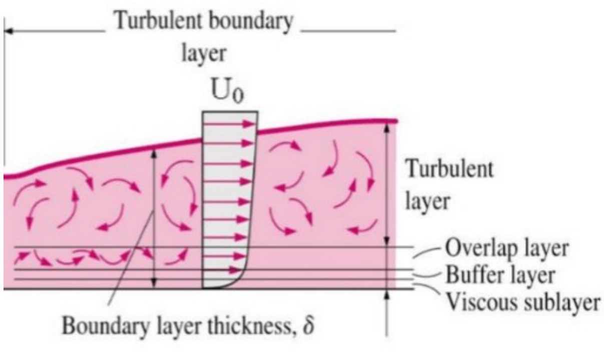


Figure 6.5: *Velocity profile close to a wall for a turbulent flow.*

A turbulent flow in a pipe is not turbulent everywhere: very close to the wall of the pipe ($r \simeq r_0$), there is a region called the viscous sub-layer, where the flow is laminar. No eddies can form in this layer and, if we could zoom in, we would see that the velocity profile follows

Re	4,000	23,000	110,000	1,100,000	3,200,000
m	1/6	1/6.6	1.7	1/8.8	1/10
$\overline{u}/u_{\max}$	1.26	1.24	1.22	1.18	1.16

Table 6.1: Coefficients of the power-law velocity profile.

the profile we found for a laminar flow. For most parts of the pipe, however, conditions are turbulent, as the thickness of the viscous sub-layer δ is very small compared to the size of the pipe:

$$\delta \ll r_0 = D/2. \tag{6.34}$$

6.5.1 Empirical function (1)

The time-average velocity for turbulent flows is often described using the following power-law function:

$$\frac{u(r)}{u_{\max}} = \left(\frac{r_0 - r}{r_0}\right)^m, \tag{6.35}$$

where m is an empirical coefficient, found by performing experiments and fitting an empirical function to the data. In this section, we will use the symbol u to denote velocity, as is often done in textbooks. Even though (6.35) does provide an accurate representation of the velocity profile, it does not predict an accurate value of the wall shear stress. A better fit in that case is the equation presented in the next subsection. Some values for the coefficients in (6.35) are given in Tab. 6.1.

6.5.2 Empirical function (2): logarithmic law of the wall

The mean shear stress in a turbulent flow is defined by an equation that resembles the one used for laminar flows (6.23):

$$\tau = \eta_t \frac{du}{dy}, \tag{6.36}$$

where η_t is now the **turbulent viscosity**. The distance y is measured from the pipe's wall:

$$y = r_0 - r. \tag{6.37}$$

In contrast to the fluid viscosity, the turbulent viscosity is not a property of the fluid but a **property of the flow** and hence depends on the solution of the equation: the turbulent viscosity is a function of the velocity and the length scales of the turbulent eddies that causes the turbulent mixing. Usually, the equation is written in terms of the kinematic turbulent viscosity $\nu_t = \eta_t/\rho$, which gives

$$\frac{\tau}{\rho} = \nu_t \frac{du}{dy}. \tag{6.38}$$

An approximation for the turbulent kinematic viscosity is given by

$$\nu_t = \kappa u_* y, \tag{6.39}$$

where the **shear or friction velocity** u_* is an approximation for the velocity of turbulent eddies and given by

$$u_* = \sqrt{\tau_0/\rho}, \tag{6.40}$$

hydraulically smooth flow	hydraulically rough flow
$k_s < \delta$	$k_s > \delta$
$y_0 = \nu/(9u_*)$	$y_0 = k_s/30$

Table 6.2: Values of the hydraulic roughness.

where τ_0 is the shear stress at the pipe's wall. The size of turbulent eddies in the vicinity of the wall is assumed to be proportional to the distance y. The constant κ is called the Von Karman constant after Theodore von Karman (1881-1963), who proved experimentally that $\kappa \simeq 0.4$. Combining the equations, we obtain

$$\frac{\tau}{\rho} = \kappa u_* y \frac{du}{dy}. \tag{6.41}$$

In the vicinity of the wall, we make the assumption that

$$\tau \simeq \tau_0 = \rho u_*^2, \tag{6.42}$$

from which we obtain

$$\frac{du}{dy} = \frac{u_*}{\kappa}\frac{1}{y}. \tag{6.43}$$

This equation can be integrated and yields

$$\frac{u}{u_*} = \frac{1}{\kappa}\ln\left(\frac{y}{y_0}\right). \tag{6.44}$$

The constant of integration y_0 is found for

$$u(y = y_0) = 0. \tag{6.45}$$

The value of this integration constant depends on the **hydraulic roughness** k_s of the wall. This roughness is compared to the thickness of the laminar viscous sub-layer δ and two cases are considered: (i) hydraulically smooth flow and (ii) hydraulically rough flow. The value of the integration constant in each case is given in Tab. 6.2.

The results of the empirical fits are compared to experimental data for hydraulically smooth flows in Fig. 6.6. In the viscous sub-layer, the velocity profile is laminar, and therefore (6.23) holds, which can be written as:

$$u(y) = \frac{\tau_0 y}{\eta} = \frac{\rho u_*^2 y}{\eta} = \frac{u_*^2 y}{\nu}, \tag{6.46}$$

leading to

$$\frac{u}{u_*} = \frac{u_* y}{\nu}. \tag{6.47}$$

On a semi-logarithmic plot, such as in Fig. 6.6, this linear profile will look curved (see the profile in the viscous sub-layer). For the logarithmic layer, using hydrodynamically smooth conditions, we get

$$\frac{u}{u_*} = \frac{1}{\kappa}\ln\left(\frac{9u_* y}{\nu}\right) = \frac{1}{\kappa}\ln(9) + \frac{1}{\kappa}\ln\left(\frac{u_* y}{\nu}\right). \tag{6.48}$$

Johann Nikuratse, a German engineer (of Georgian descent), published in 1933 a well-cited article, where he presented the study of friction that a turbulent fluid experiences as it flows through pipes. He showed that for hydrodynamically smooth conditions, he could fit his data with the function

$$\frac{u}{u_*} = 5.5 + 2.49 \ln\left(\frac{u_* y}{\nu}\right) \tag{6.49}$$

from which we can deduce that $\kappa = 0.4$, as was already observed by Von Karman. Nikuratse found for hydraulically rough flows that

$$\frac{u}{u_*} = 8.48 + 2.49 \ln\left(\frac{y}{k_s}\right). \tag{6.50}$$

By comparing this equation with the expected theoretical one,

$$\frac{u}{u_*} = \frac{1}{\kappa} \ln\left(\frac{30y}{k_s}\right) = \frac{1}{\kappa} \ln(30) + \frac{1}{\kappa} \ln\left(\frac{y}{k_s}\right), \tag{6.51}$$

we can deduce that again $\kappa = 0.4$ and we can verify that $\ln(30)/\kappa = 8.5 \simeq 8.48$.

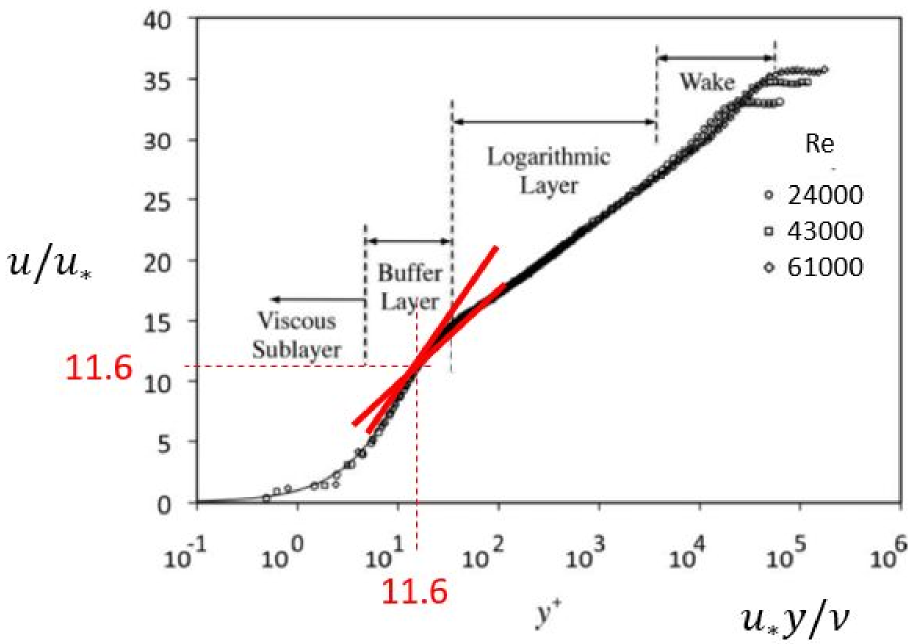


Figure 6.6: *Comparison between measured data and velocity profiles estimated for the viscous sub-layer regions and the logarithmic layer. The buffer layer and wake will not be modelled in this course.*

Note that there is a smooth transition between the viscous sub-layer and the logarithmic layer: this indicates that a single thickness of the viscous sub-layer δ does not exist. The thickness δ is therefore defined as the intersection between the slopes of the layers, as indicated in red in Fig. 6.6. It is found that

$$\delta = 11.6 \frac{\nu_t}{u_*}, \tag{6.52}$$

from which it follows that

$$y_0 = \nu_t / (9 u_*) = \frac{\delta}{105}. \tag{6.53}$$

Note from this result that y_0 is located in the laminar sub-layer. At that position, the logarithmic profile is not valid, as we have discussed. The position y_0 therefore only represents the position where the **extrapolated** logarithmic profile is zero.

6.6 The friction factor in the general case

In order to establish a relation for the friction factor in the logarithmic layer, we have to evaluate the average velocity in the pipe. This is done as follows:

$$U = \frac{1}{\pi r_0^2} \int_0^{r_0} u 2\pi r dr,$$
$$U = 2\pi \frac{u_*}{\kappa} \frac{1}{\pi r_0^2} \int_0^{r_0} (r_0 - y) \ln\left(\frac{y}{y_0}\right) dy. \tag{6.54}$$

Making use of the mathematical relation

$$\int x \ln(x) dx = \frac{1}{2} x^2 \left(\ln(x) - \frac{1}{2}\right), \tag{6.55}$$

we obtain, using the fact that $y_0 \ll r_0$,

$$\overline{u} = \frac{u_*}{\kappa} \ln\left(\exp(-3/2)\frac{r_0}{y_0}\right). \tag{6.56}$$

From the definition of the friction factor (6.16), we obtain

$$f = \frac{8\tau_0}{\rho \overline{u}^2} = \frac{8u_*^2}{\overline{u}^2}, \tag{6.57}$$

from which we can evaluate that

$$\frac{1}{\sqrt{f}} = \frac{\overline{u}}{2\sqrt{2}u_*} = \frac{1}{2\sqrt{2}\kappa} \ln\left(\exp(-3/2)\frac{r_0}{y_0}\right). \tag{6.58}$$

Using $\kappa = 0.4$ and realizing that $\ln(10)/2 \simeq 2\sqrt{2}\kappa$, we get

$$\frac{1}{\sqrt{f}} = 2\log\left(\exp(-3/2)\frac{r_0}{y_0}\right), \tag{6.59}$$

where

$$\log(x) = \frac{\ln(x)}{\ln(10)}. \tag{6.60}$$

For a hydraulically rough flow, we have seen that $y_0 = k_s/30$ and therefore

$$\frac{1}{\sqrt{f}} = 2\log\left(6.7\frac{r_0}{k_s}\right), \tag{6.61}$$

whereas for a hydraulically smooth flow, we have seen that $y_0 = \nu_t/(9u_*)$, and therefore

$$\frac{1}{\sqrt{f}} = 2\log\left(2u_*\frac{r_0}{\nu_t}\right). \tag{6.62}$$

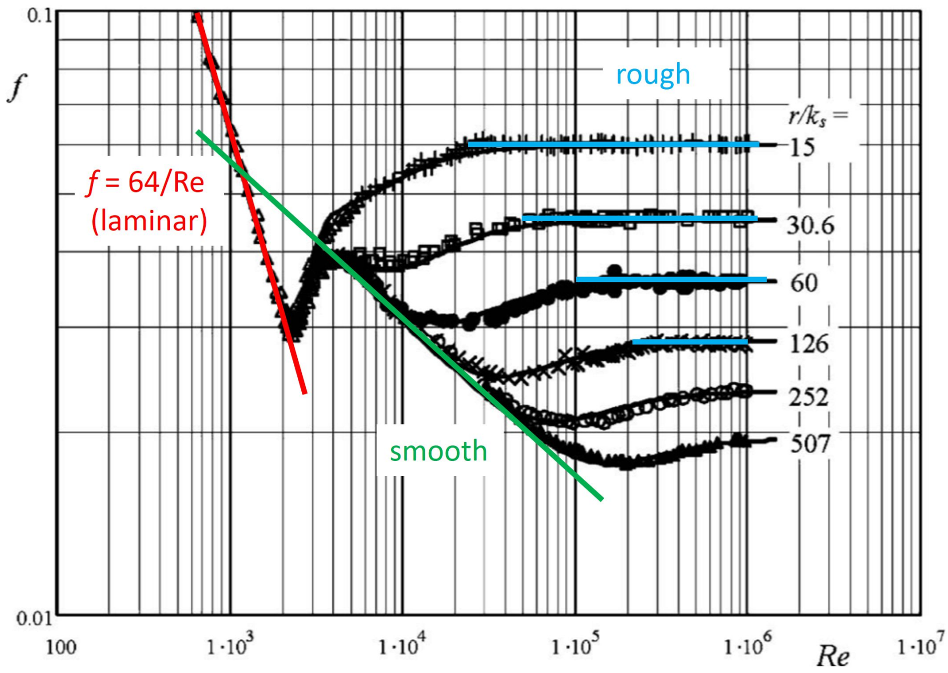


Figure 6.7: *Measurements by Nikuradse: the friction factor f as a function of Re for different values of r_0/k_s.*

Using (6.52), one gets

$$\frac{1}{\sqrt{f}} = 2\log\left(24\frac{r_0}{\delta}\right). \tag{6.63}$$

The theoretical predictions for f are compared with experimental data in Fig. 6.7.

As we have

$$\frac{r_0}{\delta} \sim \frac{r_0 u_*}{\nu_t} \sim \mathrm{Re}, \tag{6.64}$$

it follows that for a hydraulically smooth flow, f varies with Re, independently of k_s/r_0. From (6.61) it follows that for a hydraulically rough flow, f varies with k_s/r_0, independently of Re.

6.7 Flow around a rigid body

In this section, we will consider an object in a flow under the following assumptions:

1. The object under study is rigid and the fluid cannot penetrate its walls.
2. The density and viscosity of the fluid are constant.
3. The object is isolated and placed in a large amount of fluid, so that the flow of fluid far away from the object is not affected by the presence of the object.
4. The flow is steady and uniform.

The two final assumptions enable us to define the flow of the liquid far away from the object as being constant in time and direction:

$$u\,(\text{far away}) = U. \tag{6.65}$$

This implies that we do not consider a flow within a pipe, as studied in the previous sections, as we have seen that in a pipe, when friction is accounted for, the flow is dependent on the position in the cross-sectional area of the pipe. Here, we take the flow to be everywhere the same far away from the object. Close to the object, different types of flow are possible, laminar or turbulent, and different types of turbulence can be observed, as shown in Fig. 6.8

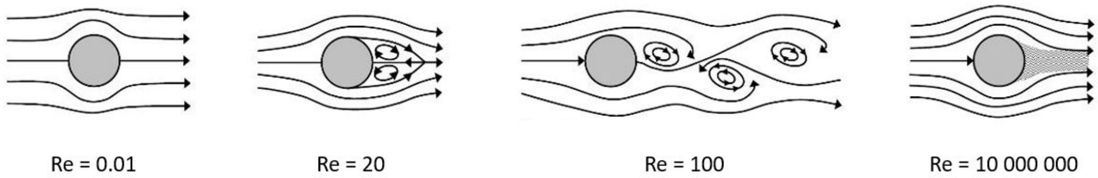


Figure 6.8: *Flow around a cylinder for different values of the Reynolds number. For Re = 0.01 the flow is laminar. In the other examples it is turbulent.*

The type of flow is related to the Reynolds number, which is given by

$$\mathrm{Re} = \frac{UL}{\nu} = \frac{\rho UL}{\eta}, \tag{6.66}$$

where L is a characteristic length for the object (e.g., its diameter).

6.7.1 Ideal fluid

In the previous Chapters, we have been setting up equations for ideal fluids, i.e., fluids without viscosity and with a constant density. In that case, $\eta = 0$, and we have

$$\mathrm{Re} = \infty. \tag{6.67}$$

In that case, we can (see exercises) apply Bernoulli's equation to find the pressure on an object:

$$P + \frac{1}{2}\rho u^2 = P_\infty + \frac{1}{2}\rho U^2, \tag{6.68}$$

where we are working at a constant altitude z. At the **stagnation point** the velocity of the fluid is zero ($u = 0$) by definition, and we deduce that

$$\Delta P = P - P_\infty = \frac{1}{2}\rho U^2 \tag{6.69}$$

This equation holds for the two stagnation points located at points 1 and 3 in Fig. 6.9. From point 1 to 2, the flow accelerates (this can be seen from the streamlines getting closer to each other), and hence the pressure difference ΔP drops, so that $\Delta P < 0$. From 2 to 3, the flow decelerates (the streamlines get farther apart again), and the pressure increases again, so that at point 3 the pressure is again equal to (6.69).

Even though there is a condition for the velocity perpendicular to the object to be zero (the fluid cannot penetrate the object), there is no condition for the tangential (parallel) component of the velocity. In an ideal fluid (not realistic), the parallel component of the velocity can have any value. This implies that for an ideal fluid there is a **slip** condition. For viscous fluids, a **no-slip condition** holds. Because the pressure force is the same at point 1 and 3, there is no net drag force on the object:

$$F_w = 0, \tag{6.70}$$

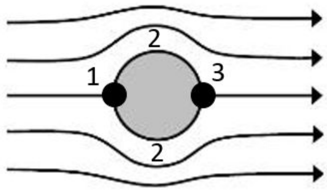


Figure 6.9: *Streamlines around a cylinder. The stagnation points are represented by points 1 and 3.*

which is a result contradicts our intuition, as our experience tells us that there *must* be a force on an object in a flow. In the next subsection we are going to study "real" liquids (with viscosity) and we will find that our intuition is correct!

We note that the velocity profile for an ideal fluid resembles the one for a flow at low Reynolds number (see Fig. 6.8), even though in this case $\mathrm{Re} = \infty$. This will also be discussed in the next subsection.

6.7.2 Viscous fluid

A very important difference between ideal and viscous fluids is that viscous fluids obey the no-slip boundary condition, which implies that both the tangential (parallel) as well as the normal (perpendicular) component of the velocity must be zero on the object:

$$\begin{aligned} u_t(\text{on the object}) &= 0, \\ u_n(\text{on the object}) &= 0. \end{aligned} \tag{6.71}$$

This has as consequence that the velocity profile in the limit $\eta \to 0$ ($\mathrm{Re} \to \infty$) will be very different from the velocity profile for an ideal fluid, for which $\eta = 0$ ($\mathrm{Re} = \infty$), as illustrated in Fig. 6.9. Indeed, because of the viscosity (despite its very small value), there will be irreversible energy losses (friction will transform kinetic energy into heat), which leads the pressure force at point 3 (as shown in Fig. 6.9) being different from the pressure force at point 1, and hence that

$$F_w \neq 0. \tag{6.72}$$

In Chapter 3 we have seen that the Reynolds number Re gives the relative magnitude of the inertial forces to the viscous forces. For small Re the inertial forces may be neglected, and for a steady flow, as assumed here, the velocity does not depend on time. This implies that the general Navier–Stokes equations reduce to the so-called Stokes equations, which we give here, for the sake of simplicity, in two dimensions and in the direction of the flow (the z-direction, which is the vertical direction, so that we also account for the weight)

$$-\rho g - \frac{\partial P}{\partial z} + \eta \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial z^2} \right) = 0. \tag{6.73}$$

This equation can be solved analytically for some important cases. In particular, Stokes solved it for the flow around a sphere. In that case, one should work in spherical coordinates (not Cartesian ones, as shown here). Deriving the Stokes equation is outside of the scope of this course, but thanks to a dimension analysis we will find the results found by Stokes (without the correct pre-factors, which we will give underneath). From (6.73), we can carry out the

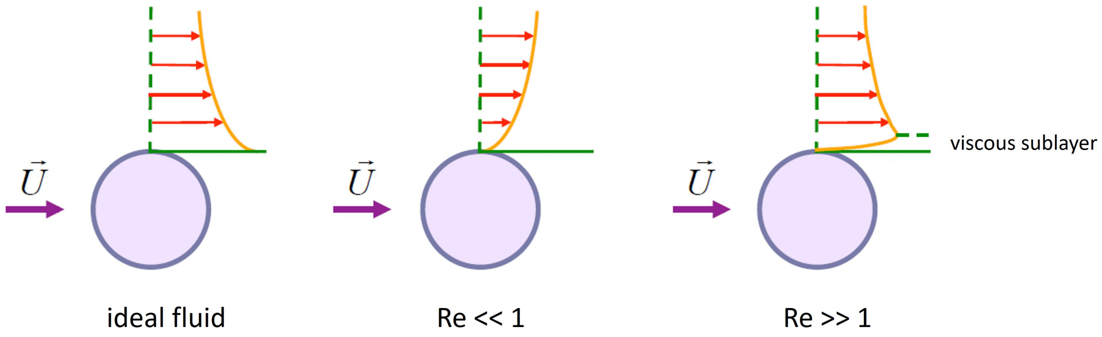


Figure 6.10: *Difference between the tangential components of the fluid velocity in the case of an ideal fluid, a fluid in the laminar regime (Re $\ll 1$) and a fluid in the turbulent regime (Re $\gg 1$). Note that close to the object, a small laminar region can always be found. For Re $\ll 1$ the flow is laminar everywhere, and for Re $\gg 1$ the flow is laminar in the region called the viscous sub-layer.*

following dimension analysis, using L as the symbol for the dimension of a length and U as the symbol for velocity:

$$F_g = mg == \rho g L^3,$$
$$F_p == \frac{P}{L} L^3 = PL^2,$$
$$F_w == \eta \frac{U}{L^2} L^3, = \eta U L \tag{6.74}$$

where F_g is the weight, which is proportional to mass m multiplied by the gravity constant g. The L^3 comes from the fact (see Chapter 3) that the Navier–Stokes equations are the Newton's second law divided by the volume of the considered volume element. We find, as expected, that the pressure force is proportional to pressure multiplied by a surface area ($S = L^2$) and that the drag force is proportional to $\eta U L$.

Usually, the drag force is expressed as

$$F_w = C_d \frac{1}{2} \rho U^2 A, \tag{6.75}$$

where C_d is called the **drag coefficient** and A is the the reference area of the object The reference area is the area of the projected frontal area of the object. For a sphere, for example, $A = \pi r^2$, where r is the radius of the sphere, as illustrated in Fig. 6.11.

By dimension analysis, comparing (6.74) and (6.75), we find that

$$C_d = \frac{\eta}{\rho U L} = \frac{1}{\mathrm{Re}}. \tag{6.76}$$

Stokes found that for a sphere

$$C_d = \frac{24}{\mathrm{Re}}. \tag{6.77}$$

From this result, we can evaluate the drag force on a sphere, using $L = 2r$ and $A = \pi r^2$

$$F_w = 6 \eta \pi r U. \tag{6.78}$$

This is a result that is convenient to remember, as this drag force is used a great deal in hydraulic engineering: using this drag force the settling velocity of sand and silt particles can

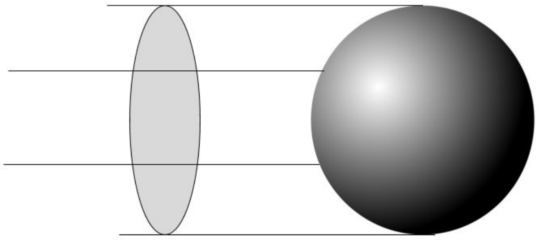


Figure 6.11: *Definition of the reference area for a sphere in a steady uniform flow, which corresponds to the projected area.*

be estimated, which is at the core of numerical simulation models for sediment transport. We have to be careful here: in the derivation presented above we have implicitly assumed that we study the drag force on an object *at rest* in a moving flow, when in reality we consider the drag on a *moving* sand or silt particle in a fluid at rest. How can we relate these two situations? This is quite simple. We only need to perform a change of the reference frame.
Case (I): when the fluid is moving far away from the object, which is at rest:

$$\begin{aligned} u_{\text{fluid}} &= U, \\ u_{\text{object}} &= 0. \end{aligned} \tag{6.79}$$

Case (II): when the fluid is at rest far away from the object, which is moving:

$$\begin{aligned} u_{\text{fluid}} &= 0, \\ u_{\text{object}} &= U_0. \end{aligned} \tag{6.80}$$

If we now change frame of reference and use the frame of reference of the object, we get

$$\begin{aligned} u_{\text{fluid}} &= -U_0, \\ u_{\text{object}} &= 0, \end{aligned} \tag{6.81}$$

and we can therefore apply the results using $U = -U_0$.

One important point of attention is the direction of the vector $\vec{F_w}$. The drag force on an object **always opposes its movement**. This means that if the object moves to the right, the drag force will be oriented to the left and vice-versa. Therefore

$$\vec{F_w} = -6\eta\pi r\vec{U_0}. \tag{6.82}$$

The energy lost by friction can be estimated by

$$\mathcal{P} = F_w U. \tag{6.83}$$

The drag coefficient of an object depends on the Reynolds number. Equation (6.77) is valid for a laminar flow (low Re). The behaviour of the drag coefficient as a function of Reynolds number for a sphere is given in Fig. 6.12.

6.8 Summary

After studying this chapter you should be able to:

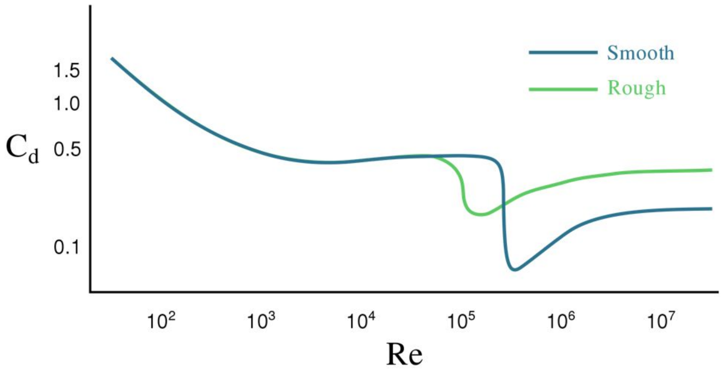


Figure 6.12: *Drag coefficient as a function of Reynolds number for a sphere with either a rough or a smooth surface.*

1. Know which parameters the behaviour of a flow depends on and how these parameters are accounted for in the Reynolds number; know the (approximate) Reynolds number range that defines a laminar and a turbulent flow.
2. Understand the concept of developing and fully developed flows.
3. Know the definitions of pipe head loss and component head loss.
4. Be able to define the friction factor f.
5. Be able to set up the Darcy–Weisbach equation and apply it.
6. Be able to derive the velocity profile for a laminar flow in a pipe (Poiseuille flow).
7. Be able to define the velocity profile of a turbulent flow using the concepts of viscous sub-layer and turbulent layer; understand the definition of a logarithmic-law profile, turbulent viscosity, shear velocity, Von Karman constant and hydraulic roughness.
8. Know the difference between slip and no-slip conditions.
9. Be able to define the drag coefficient C_d and relate it to the drag force using dimension analysis.
10. Know the drag force on a sphere, i.e.,

$$F_w = 6\eta\pi r U.$$

The Dutch corner

English	Nederlands
rigid body	star lichaam
drag force	sleepkracht
lift force	liftkracht
stagnation point	stuwpunt

7 Gradually varied flow

7.1 Introduction

In this Chapter, we are going to review the equations that are relevant to describe gradually varying flows in pipes and open channels, which involves the study of flows on slopes. An engineer often has to estimate the water-surface profile for a given discharge. For example, when a dam has to be designed, it is important to know the water-surface profile in the river upstream in order to define the amount of land that will be required for the upstream reservoir.

As we have seen in Chapters 5 and 6, gradually varying flow in a pipe is easy to describe using the balance equations. For open channels, the flow is in contact with the atmosphere at the top (the channel is "open"), and as we know from Chapter 5 the water-air interface has to adapt to the flow conditions.

7.2 Friction slope i_f, bottom slope i_b and water surface slope i_w

We consider a mildly sloping channel as presented in Fig. 7.1. Three important slopes are defined.

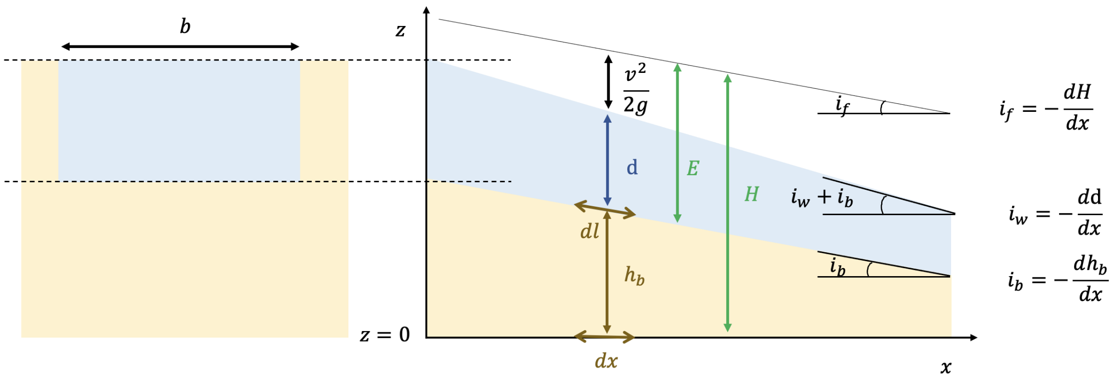


Figure 7.1: *Left: cross-section of a rectangular channel. Right: transect of the same channel.*

7.2.1 Bottom slope

Let's start by defining the bottom slope i_b. The **bottom slope is simply the slope of the bottom of the channel**, over which the water is flowing:

$$i_b = -\frac{dh_b}{dx}, \tag{7.1}$$

where $z = h_b(x)$ is the vertical position of the bottom of the channel at position x. The minus signs arises from the fact that the parameter i_b is defined as a positive quantity when the slope is negative in the x-direction. Because slopes in all the applications we will consider are small, we have (from the small-angle approximation)

$$\frac{dh_b}{dx} = \frac{dh_b}{dl}, \tag{7.2}$$

where l is the distance along the slope. For small slopes, i_b is also equal to the slope angle, as indicated in Fig. 7.1.

7.2.2 Water-surface slope

The water-surface slope i_w is defined as

$$i_w = \frac{d\mathrm{d}}{dx}. \tag{7.3}$$

For small slopes i_w is also given by the angle indicated in Fig. 7.1.

7.2.3 Friction slope

The friction slope is defined as the rate at which energy is lost due to friction along a channel, so for a system experiencing *only friction loss* the friction slope can be written as

$$i_f = -\frac{dH}{dx}. \tag{7.4}$$

We note that i_b, i_w and i_f are dimensionless (they have a dimension of a length divided by a length).

7.2.4 Notation

To designate friction slope and bottom or channel slope there is no uniform symbolic convention. In these lecture notes, the notation typically used in the department of Hydraulic Engineering of the TU Delft is adopted. In textbooks from the United States, the friction slope i_f is usually designated by S_f (where S stands for slope) and the bottom or channel slope i_b by S_0. In the present lecture notes, S designates a surface area. The water-surface profile is designated by dy/dx in these books.

7.3 Friction slope in a pipe

We are now going to define the friction slope (frictional gradient) i_f based on the results obtained in Chapter 6 for the flow in pipes. The flow in pipes is symmetric around the central axis of the pipe, as the friction is exerted on the wall of the pipe, which leads to a rotational symmetry. The only energy loss is due to friction, and there is no energy gain. We can therefore write

$$\left(\frac{P_{\text{in}}}{\rho g} + z_{\text{in}}\right) - \left(\frac{P_{\text{out}}}{\rho g} + z_{\text{out}}\right) = h_f. \tag{7.5}$$

In fact, this equation can also be written as (recalling that $v_{\text{in}} = v_{\text{out}}$)

$$\left(\frac{v_{\text{in}}^2}{2g} + \frac{P_{\text{in}}}{\rho g} + z_{\text{in}}\right) - \left(\frac{v_{\text{out}}^2}{2g} + \frac{P_{\text{out}}}{\rho g} + z_{\text{out}}\right) = h_f. \tag{7.6}$$

This enables us to write

$$(H_{\text{out}} - H_{\text{in}}) = (h_{\text{out}} - h_{\text{in}}) = -h_f. \tag{7.7}$$

We found in Chapter 6 that

$$h_f = \frac{\tau_0 A}{\rho g S} = \frac{4\tau_0}{\rho g}\frac{L}{D}. \tag{7.8}$$

We are now going to write these equations in a different form. Using the same procedure we used to evaluate the laminar flows in pipes, we will now consider a small tube of length dl (instead of L). This enables to change (7.7) and (7.8) into

$$-\,dH = h_f, \tag{7.9}$$

$$h_f = \frac{4\tau_0}{\rho g}\frac{dl}{D}. \tag{7.10}$$

Doing so, the only change we made was to define

$$dH = H_{\text{out}} - H_{\text{in}}. \tag{7.11}$$

The small d indicates that we have a small variation (for a large variation one usually uses Δ). By definition a difference is always “what comes out minus what comes in”, which leads to the minus sign in (7.9). Combining the two equations gives

$$i_f = -\frac{dH}{dl} = -\frac{dh}{dl} = \frac{h_f}{dl} = \frac{4\tau_0}{\rho g D}. \tag{7.12}$$

The parameter i_f represents the amount of head loss per unit length of pipe. For a pipe of length L having a constant frictional gradient, we obtain for a circular pipe

$$i_f = -\frac{\Delta H}{L} = -\frac{\Delta h}{L} = \frac{h_f}{L} = \frac{4\tau_0}{\rho g D}. \tag{7.13}$$

Using the Darcy–Weisbach equation, we get

$$i_f = f\frac{v^2}{2gD}. \tag{7.14}$$

This equation is an alternative form of the Darcy–Weisbach equation.

7.4 Hydraulic radius

Channels can have different shapes and therefore different length scales. We have so far discussed the length and width of a channel, but if the channel has an irregular shape (such as a trapezoidal channel, see Figure 7.3) the equations should be adapted to this. A convenient way to deal with the changes in geometry is to use the concept of hydraulic radius that we are going to introduce now. It is then simply a matter of using the right hydraulic radius in the equations to deal with different geometries.

By definition, we have

$$i_f = \frac{h_f}{L}. \tag{7.15}$$

We recall, from Chapter 6, the result obtained when we derived the Darcy–Weisbach equation for a pipe:

$$h_f = \frac{\tau_0 A}{\rho g S}, \tag{7.16}$$

where $A = \pi DL$ is the surface area over which the wall shear stress is acting, S is the cross-sectional area of the flow and L the length over which h_f (and hence i_f) is evaluated. We now rewrite this equation as

$$i_f = \frac{\tau_0 A}{\rho g S L} = \frac{\tau_0}{\rho g R}. \tag{7.17}$$

The **hydraulic radius** R is a quantity that can be both defined for a pipe and an open channel. It is defined as the cross-sectional area of the channel or pipe divided by the **wetted perimeter**:

$$R = \frac{\text{cross-sectional area}}{\text{wetted perimeter}}. \tag{7.18}$$

The wetted perimeter includes all external surfaces the fluid exerts shear stress upon. In a pipe, the fluid is in contact with the wall of the pipe (where the shear stress τ_0 is acting). The wetted perimeter can thus be defined as $\pi D = 2\pi r_0$. The cross-sectional area of a pipe is given by $S = \pi r_0^2$. For a pipe filled with water, we therefore have

$$R = \frac{\pi r_0^2}{2\pi r_0} = \frac{r_0}{2}. \tag{7.19}$$

We recall that we found for laminar (Poiseuille) flows that

$$\tau(r) = -\frac{r}{2}\rho g \frac{dh}{dl}, \tag{7.20}$$

and that $dh/dl = -h_f/dl$, from which we deduce that

$$i_f = \frac{2\tau(r)}{\rho g r}. \tag{7.21}$$

As i_f is constant over the cross-section of the pipe, we get

$$i_f = \frac{2\tau_0}{\rho g r_0} = \frac{2\tau(r)}{\rho g r}, \tag{7.22}$$

$$\tau(r) = \tau_0 \frac{r}{r_0}. \tag{7.23}$$

The shear stress is increasing linearly from $r = 0$ to $r = r_0$, as illustrated in Fig. 7.2. This result has been derived under the assumption of laminar flow. However, it is also valid for some turbulent flows.

This result can be generalised for different geometries: the hydraulic radius R for other common geometries are given in Fig. 7.3 and Tab. 7.1.

7.4.1 Hydraulic radius in pipe flow

For laminar (Poiseuille) flow, the mean velocity across a pipe is given by

$$\overline{v} = -\frac{r_0^2}{8\eta}\rho g \frac{dh}{dl}, \tag{7.24}$$

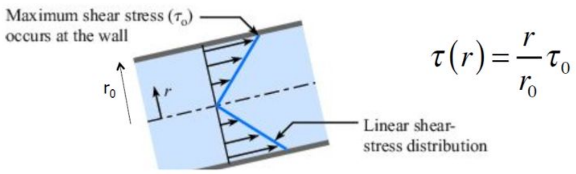


Figure 7.2: *Profile of shear stress in a pipe for laminar flow.*

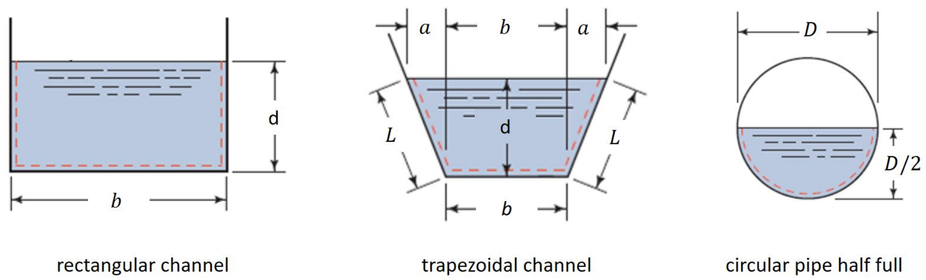


Figure 7.3: Common open-channel geometries.

	Rectangular channel	Trapezoidal channel	Half-full circular pipe
Cross-sectional area	$b\mathrm{d}$	$b\mathrm{d} + a\mathrm{d}$	$\pi D^2/8$
Wetted perimeter	$b + 2\mathrm{d}$	$b + 2L$	$\pi D/2$
Hydraulic radius R	$b\mathrm{d}/\,(b + 2\mathrm{d})$	$(b\mathrm{d} + a\mathrm{d})\,/\,(b + 2L)$	$D/4$

Table 7.1: The hydraulic radius for common open-channel geometries.

which can be written as function of i_f

$$i_f = \frac{8\eta\overline{v}}{\rho g r_0^2}. \tag{7.25}$$

In pipe flows studies for water, at room temperature, the mean velocity of water is given as function of the hydraulic radius and the friction slope by the Hazen–Williams equation

$$\overline{v} = 0.849 C R^{0.63} i_f^{0.54}. \tag{7.26}$$

As one may suspect from the coefficients, this equation is empirical. In contrast to the Poiseuille flow (limited to flows at small velocities in small pipes), this equation correctly describes the turbulent flow in pipes. The roughness coefficient C is a dimensionless parameter. The number 0.849 has dimension of $\mathrm{m}^{0.37}/\mathrm{s}$, and the hydraulic radius should be measured in m. The coefficient C is dimensionless and lies in the range 100 - 150 for most materials. This equation can be compared to the Darcy–Weisbach equation (7.14). Realizing that the hydraulic radius for a pipe is given by $R = D/4$, the Darcy–Weisbach can be rewritten

$$\overline{v} = \left(\frac{8gR}{f}\right)^{1/2} i_f^{1/2}. \tag{7.27}$$

One can see that the exponent for i_f is indeed close to 1/2 in the empirical formulation (7.26). The power of the dependence on hydraulic radius is predicted to be 1/2 using the Darcy–Weisbach equation, and is found to be 0.63 in realistic flows. Nothing very conclusive can be said however, as the dependence of the friction coefficient f and the roughness parameter C on R is unknown. Both these coefficients are related to the wall shear stress τ_0, which depends on the roughness of the pipe. This roughness also influences the hydraulic radius, as it changes the wetted perimeter.

For a Poiseuille flow, we have found in Chapter 6 (using $R = D/4$) that

$$f = \frac{16\eta}{\rho R\overline{v}}. \tag{7.28}$$

Using the Darcy–Weisbach equation, we can show that

$$i_f = \frac{\Delta h}{L} = \frac{2\eta}{\rho R^2}\frac{\overline{v}}{g}, \tag{7.29}$$

from which we find that, using $Q = S\overline{v}$,

$$Q = \frac{\rho g R^2}{2\eta} S \frac{\Delta h}{L}. \tag{7.30}$$

This last equation is known as the **Darcy equation**, and the coefficient

$$K = \frac{\rho g R^2}{2\eta} \tag{7.31}$$

is called the **hydraulic conductivity**. In general form, the Darcy equation is given by

$$Q = KS\frac{\Delta h}{L}. \tag{7.32}$$

The hydraulic conductivity is a coefficient that can be estimated by measuring the volume flux through a medium of cross-sectional area S and of length L, as well as the change in piezometric heads along the pipe. A similar equation is extremely important in the study of soils. In that case, it can be shown that K becomes a function of the permeability of soils (i.e., their capacity to allow water to pass through).

7.4.2 Hydraulic radius in open-channel flow

In open-channel flow, the mean fluid velocity can be linked to the hydraulic radius according to the Chézy equation, which was derived in 1775:

$$\overline{v} = C\sqrt{R\, i_b}, \tag{7.33}$$

where C is called the Chézy coefficient. This formulation has been adapted by the Irish professor Robert Manning in 1890 (the adaptation had already been discovered by the French engineer Gauckler in 1867). The Chézy equation has been used to build canals in projects all around the world, for instance during the construction of the Panama canal. Gauckler and Manning proposed

$$C = \frac{1}{n}R^{1/6}, \tag{7.34}$$

where n is called Manning's roughness coefficient (with the unusual units of s/m$^{1/3}$). In principle, using Manning's equation is equivalent to the Chézy equation (both depend on an empirical parameter!), but the Manning formula is more widely accepted in the United States. Another formula, called GMS (for Gauckler–Manning–Strickler) is given by

$$\overline{v} = K_s R^{2/3}\sqrt{i_b}, \tag{7.35}$$

where one can easily show that the Strickler coefficient and the Manning coefficient are linked by

$$K_s = \frac{1}{n}, \tag{7.36}$$

The Strickler coefficient K_s varies between 20 (stone) and 80 (smooth concrete). Contrary to what you have been taught, it has become common in practise to present these coefficient values without units. It is very important therefore to always use values of the hydraulic radius in m and the velocity in m/s. Strickler was a German engineer, who worked to show the relationship between n and the mean particle size of sediment. He showed that

$$n = \frac{D_{50}^{1/6}}{21}, \tag{7.37}$$

where D_{50} is the mean particle size of sediment found in river beds.

The Chézy equation is very similar to the Darcy–Weisbach equation discussed in the previous subsection. A notable difference is that the Chézy equation is a function of i_b and not i_f.The reason is that, quite simply, Chézy was unaware of the concept of i_f and the only variable he could relate to was the (measurable) i_b!. The Chézy equation and similar equations (Manning, GMS) are in fact applicable to uniform (quasi-)steady open-channel flows, for which $i_f = i_b$. In this type of flow, the water depth d is constant and hence so is the velocity U of the fluid. It is then possible to write

$$i_b = -\frac{dh_b}{dl} = -\frac{d\left(d + \frac{U^2}{2g} + h_b\right)}{dl} = -\frac{dH}{dl} = i_f. \tag{7.38}$$

In order for uniform steady flow conditions to apply, the flow must be able to adjust relatively quickly to changes in flow conditions. In other words, the time scale of adjustment must be much shorter than the time scale associated with the changes to flow conditions. For example, for a tidal channel with $U = 1$ m/s and $d = 10$ m, the adaptation time scale would be extremely short compared to variations in the driving tide with a period of 12 h. The steady-flow assumption is therefore justified.

7.5 Logarithmic law of the wall

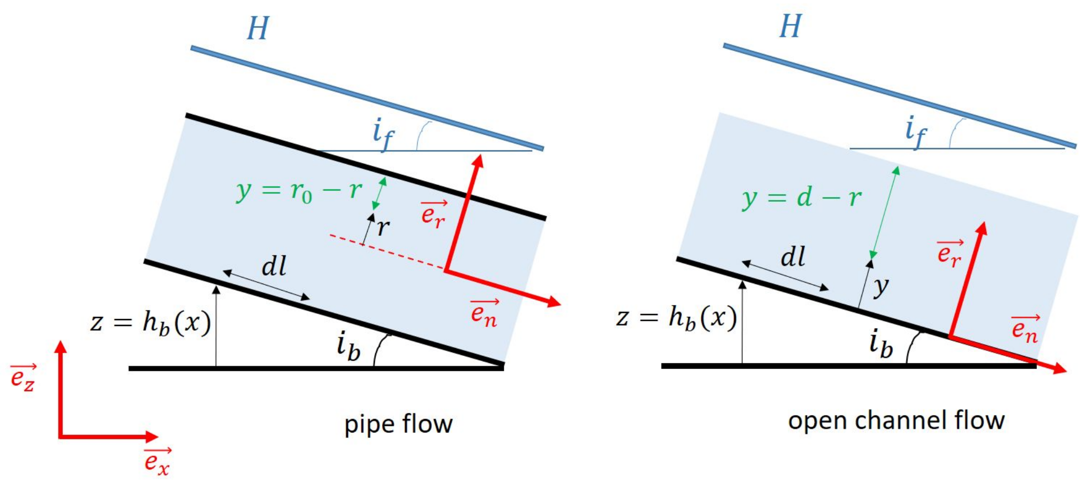


Figure 7.4: *Comparison between the coordinate systems to study pipe flow and open-channel flow; in the case of the open channel the flow is uniform and steady.*

Using the same arguments as those presented in Chapter 6 for the flow through pipes, we find that

$$\tau(y) = \rho g i_b \left(\mathrm{d} - y\right) = \tau_b \left(1 - \frac{y}{\mathrm{d}}\right), \tag{7.39}$$

where τ_b is the shear stress at the bed (where $y = 0$). The shear therefore decreases linearly from $\tau(y = 0) = \tau_b$ to $\tau(y = \mathrm{d}) = 0$. This linear dependence is the same as the one we found in pipes. If the flow is laminar, the flow profile will be parabolic, as we found for a Poiseuille flow in a pipe. However, flows in hydraulic engineering are never laminar, but rather turbulent. For a turbulent flow in a pipe, we found that a logarithmic profile can be derived. It is questionable whether the turbulence in an axisymmetric pipe would be the same as in an open channel as studied here. In this course, we will proceed under this assumption. Therefore, when the flow is turbulent, we will assume that

$$\frac{u}{u_*} = \frac{1}{\kappa} \ln\left(\frac{y}{y_0}\right). \tag{7.40}$$

This equation can be integrated over the depth d, and the average velocity U is then given by (see exercises)

$$\frac{U}{u_*} = \frac{1}{\kappa}\left(\ln\left(\frac{\mathrm{d}}{y_0}\right) - 1\right) = \frac{1}{\kappa} \ln\left(\exp(-1)\frac{\mathrm{d}}{y_0}\right). \tag{7.41}$$

The following **friction coefficient** C_f is usually defined

$$\tau_b = C_f \rho U^2 = \rho u_*^2, \tag{7.42}$$

where

$$\frac{1}{\sqrt{C_f}} = \frac{1}{\kappa} \ln\left(\exp(-1)\frac{\mathrm{d}}{y_0}\right). \tag{7.43}$$

(As a side note: one can observe that for flows in pipes the friction factor f and the friction coefficient C_f are linked by $C_f = f/8$.) It follows that

$$i_f = \frac{\tau_b}{\rho g R} = \frac{C_f U^2}{gR}. \tag{7.44}$$

Let's consider a rectangular channel with a very large width b. In that case, the hydraulic radius can be estimated to be equal to

$$R = \frac{b\mathrm{d}}{b + 2\mathrm{d}} \simeq \mathrm{d}, \tag{7.45}$$

from which follows that

$$i_f = i_b = C_f \frac{U^2}{g\mathrm{d}}. \tag{7.46}$$

The friction coefficient C_f can then be estimated, as i_f is equal to i_b and the volume flux can be estimated, from which U can be derived. One can observe the similarities between this equation and the one derived by Chézy, which yields

$$i_f = i_b = C^{-2} \frac{U^2}{\mathrm{d}}. \tag{7.47}$$

There is therefore a simple relation between C_f and C: $C_f = gC^{-2}$. The friction coefficient (C_f or C) can therefore be estimated from the volume flux and bed slope. The volume flux is usually expressed as

$$q = \frac{Q}{b} = U\mathrm{d}, \tag{7.48}$$

leading to

$$i_f = i_b = C_f \frac{q^2}{g\mathrm{d}^3}. \tag{7.49}$$

7.6 Gradually varied flows

Using the definition of the energy E given in Chapter 5, we can write that

$$H = h_b + E. \tag{7.50}$$

From the definitions (7.1) and (7.4)) it follows that

$$\frac{dE}{dx} = i_b - i_f. \tag{7.51}$$

Doing so, we recall that we assume that the flow only experiences friction losses, so that we can assume that $h_f = -dH$. For a constant volume flux q per unit width, the energy E only depends on the local depth d:

$$E = d + \frac{q^2}{2g\mathrm{d}^2}. \tag{7.52}$$

Mathematically, we can write

$$\frac{dE}{dx} = \frac{dE}{d\mathrm{d}} \frac{d\mathrm{d}}{dx}. \tag{7.53}$$

From the taking the derivative, we find

$$\frac{dE}{d\mathrm{d}} = 1 - \mathrm{Fr}^2, \tag{7.54}$$

where we recall (from Chapter 5) that

$$\mathrm{Fr}^2 = \frac{q^2}{g\mathrm{d}^3}. \tag{7.55}$$

From combining these equations, we obtain a very important equation, which expresses the water-surface profile as a function of the bed profile:

$$\frac{d\mathrm{d}}{dx} = -i_w = \frac{i_b - i_f}{1 - \mathrm{Fr}^2}. \tag{7.56}$$

This equation is called the **Bélanger equation**, after Jean-Baptiste Bélanger (1790-1874), who in 1828 proposed this equation also known as the **backwater equation** for steady one-dimensional gradually varied flows in an open channel. In the exercises, we will see how this equation can be obtained from the balance of forces on a appropriately chosen Eulerian control volume. We will see in the next section that from this equation, if i_b, i_f and Fr are known, the water surface profile $\mathrm{d}(x)$ can be calculated.

7.7 Solving the Bélanger equation

In this section, we start by solving the Bélanger equation for steady uniform flows and then proceed to a more general case.

7.7.1 Steady uniform flows

We start by considering a steady uniform flow. In that case we have seen, see (7.38) that

$$i_b = i_f, \tag{7.57}$$

from which, using (7.56), we find that

$$\frac{d\mathrm{d}}{dx} = -i_w = 0. \tag{7.58}$$

For a steady uniform flow, the water-surface profile is not changing as function of distance x. For this type of flow $\mathrm{d} = \mathrm{d}_e$, where d_e is called the equilibrium depth (also called normal depth in textbooks from the United States).

This equation tells us that for uniform steady flows the force of gravity (represented by i_b) is in equilibrium with the friction force (represented by the term i_f). When this happens, the water depth is at equilibrium ($i_w = 0$), and this equilibrium water depth is given by, using (7.49),

$$\mathrm{d} = \mathrm{d}_e = \left(\frac{C_f}{i_b}\frac{q^2}{g}\right)^{1/3}. \tag{7.59}$$

7.7.2 The general case

The friction coefficient is given by

$$i_f = \frac{C_f U^2}{gR}. \tag{7.60}$$

The Froude number is given by

$$\mathrm{Fr} = \frac{U}{\sqrt{g\mathrm{d}}} = \frac{q}{\sqrt{g\mathrm{d}^3}}. \tag{7.61}$$

The Bélanger equation can therefore be written as

$$\frac{d\mathrm{d}}{dx} = -i_w = \frac{i_b - C_f U^2/(gR)}{1 - U^2/(g\mathrm{d})}. \tag{7.62}$$

When the width of the channel is very wide ($R \simeq \mathrm{d}$), the equation can be simplified into, using $q = U\mathrm{d}$,

$$i_w = -i_b \frac{\mathrm{d}^3 - C_f q^2/(i_b g)}{\mathrm{d}^3 - q^2/g}, \tag{7.63}$$

and we obtain

$$i_w = i_b \frac{\mathrm{d}^3 - \mathrm{d}_e^3}{\mathrm{d}^3 - \mathrm{d}_c^3}, \tag{7.64}$$

where d_c is the **critical depth** defined in Chapter 5, which corresponds to the depth when Fr $= 1$, and d_e is given by (7.59). Note that

$$\mathrm{d}_e = \left(\frac{C_f}{i_b}\right)^{1/3} \mathrm{d}_c, \tag{7.65}$$

$$\mathrm{Fr}^2 = \frac{\mathrm{d}_c^3}{\mathrm{d}^3}. \tag{7.66}$$

The Bélanger equation is a first-order ordinary differential equation and therefore requires one boundary condition (i.e., the value of d at a given location) to be solved. At the far end of the channel, we will assume that the flow is uniform and steady, and at that location the water depth is defined by $\mathrm{d} = \mathrm{d}_e$ (and, consequently, $i_w = 0$).

The Bélanger equation is usually solved numerically, as it is then possible to evaluate it for complex channel geometries. In this course, we limit ourselves to very simple geometries, and we will only discuss an analytical solution of the Bélanger equation in one of the following subsections. First, we will classify the possible solutions into different types.

7.7.3 Types of water-surface profiles

Water-surface profiles are divided into different types and defined by a name, which consists of:

- A letter, which refers to the value of i_b and indicates if the slope is mild (M) or steep (S). To know whether the slope is mild or steep, one should compare the **critical flow depth** d_c to the **uniform flow depth** d_e. The uniform flow depth d_e (usually called the **equilibrium water depth**) is the depth the water would have if the flow would be uniform and steady. If $\mathrm{d}_e > \mathrm{d}_c$, then the slope is mild (M), whereas when $\mathrm{d}_e < \mathrm{d}_c$ the slope is steep (S). When $\mathrm{d}_e = \mathrm{d}_c$ the slope is said to be critical (C). Two other letters (A and H) are used for very specific flows called adverse slope and horizontal slope, but these will not be discussed in this course.

- A number (1, 2 or 3), which refers to the actual water surface in relation to the position of the water surface for uniform and critical flow. If the water surface is above those for uniform and critical flow then the flow is a type 1. If the water surface is in between those for uniform and critical flow then the flow is a type 2. If the water surface is below those for uniform and critical flow then the flow is a type 3.

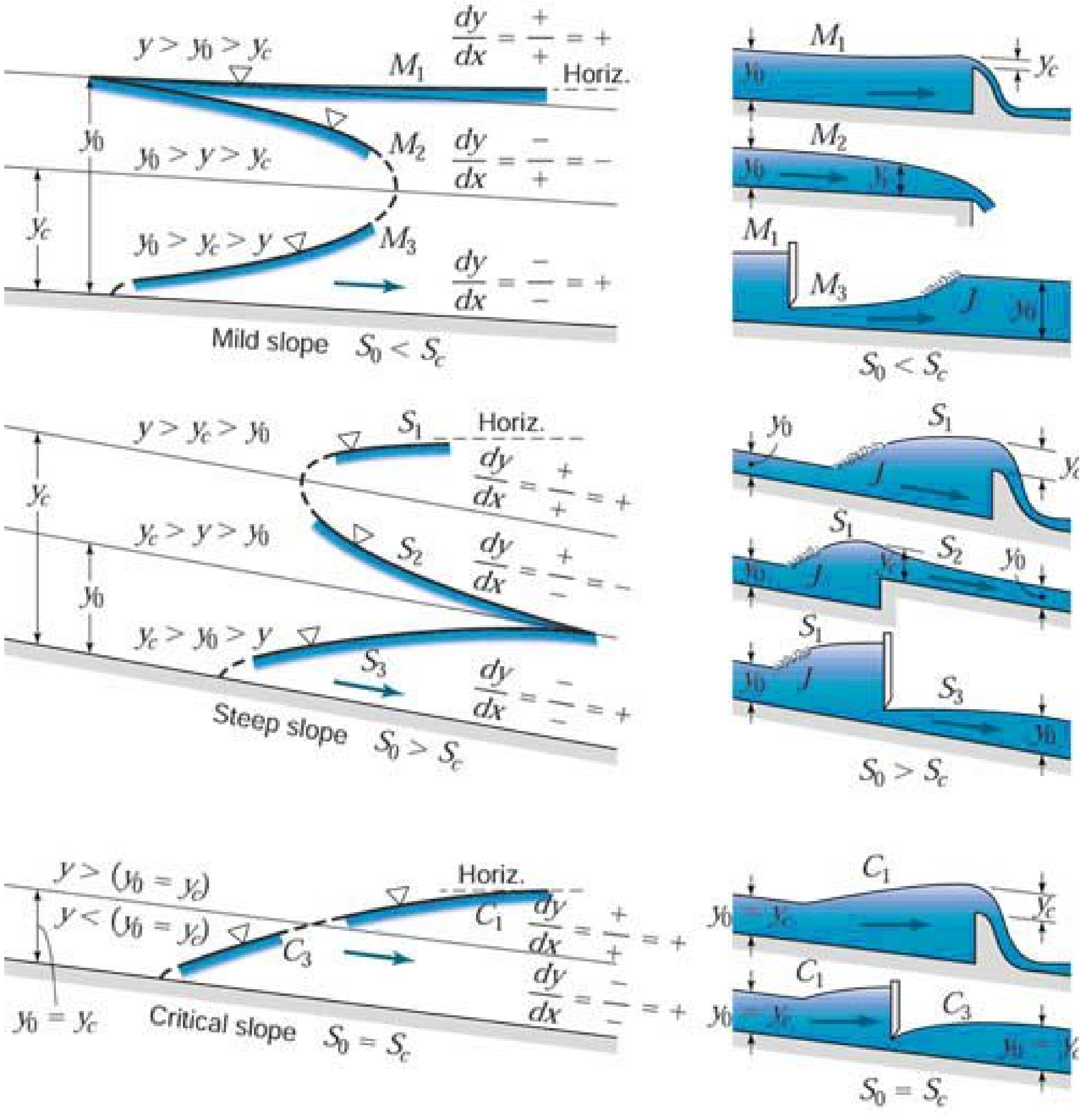


Figure 7.5: *Different types of slopes with y denoting the depth, y_0 the equilibrium depth d_e, and y_c the critical depth d_c.*

The Fig. 7.5 distinguishes the following six types of profiles known as **backwater curves**:

- An M1 water-surface profile is a profile for which $d > d_e > d_c$. In that case, one finds that $i_w > 0$, which implies that the water depth increases along the flow direction, and therefore the velocity decreases (as the flow rate remains constant). The flow is sub-critical as $d > d_c$ and hence $Fr < 1$.

- An M2 water-surface profile is a profile for which $d_e > d > d_c$. In that case, one finds that $i_w < 0$, which implies that the water depth decreases along the flow direction, and therefore the velocity increases (as the flow rate remains constant). The flow is sub-critical as d > d_c and hence Fr < 1.

- An M3 water-surface profile is a profile for which d_e > d_c > d. In that case, one finds that $i_w > 0$, which implies that the water depth increases along the flow direction, and therefore the water velocity decreases (as the flow rate remains constant). The flow is super-critical as d < d_c and hence Fr > 1.

- A S1 water-surface profile is a profile for which d > d_c > d_e. In that case, one finds that $i_w > 0$,which implies that the water depth increases along the flow direction, and therefore the velocity decreases (as the flow rate remains constant). The flow is sub-critical as d > d_c and hence Fr < 1.

- A S2 water-surface profile is a profile for which d_c > d > d_e. In that case, one finds that $i_w < 0$, which implies that the water depth decreases along the flow direction, and therefore the velocity increases (as the flow rate remains constant). The flow is super-critical as d < d_c and hence Fr > 1.

- A S3 water-surface profile is a profile for which d_c > d_e > d. In that case, one finds that $i_w > 0$, which implies that the water depth increases along the flow direction, and therefore the water velocity decreases (as the flow rate remains constant). The flow is super-critical as d < d_c and hence Fr > 1.

It is important to realize that the value of the water depth at a point determines the flow in one specific direction:

A super-critical flow (torrential motion, Fr > 1) is determined by an upstream boundary condition, as a change in water depth will affect only the flow downstream from that point (in an extreme example, if you put your hand in a water fall, the only portion of the flow affected by that action will be the flow of water below your hand). The upstream water levels are unaffected by downstream control. The transition from super-critical to sub-critical flow can only occur by means of a hydraulic jump. The position of the jump depends on the boundary conditions, but not on the presence of a discontinuity in the bed. The Bélanger equation is not applicable at the position of the hydraulic jump, as significant energy is dissipated locally, but it is in the portions of the flow upstream and downstream from the jump.

A sub-critical flow (fluvial motion, Fr < 1) is determined by a downstream boundary condition, as a change in water depth will affect the flow upstream from that point (if you put your hand in a water stream flowing gently in a small canal, you will see ripples form upstream from your hand). The upstream water levels are affected by downstream control. The transition from sub-critical to super-critical flow can only occur at a discontinuity in the channel (such as a bend, a weir or a spillway). The position of the discontinuity is fixed. This is called a **control section**.

When the Froude number is close to 1, as discussed in Chapter 5, the flow conditions tend to become unstable, resulting in wave formation.

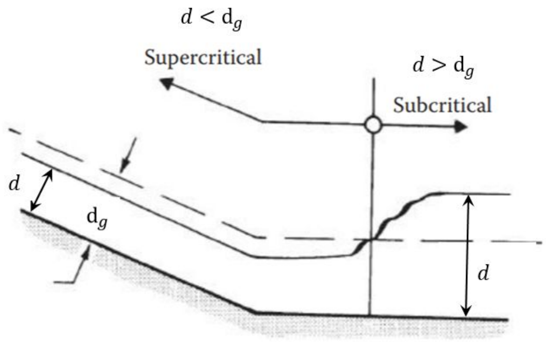


Figure 7.6: *The super-critical flow (on the left) is determined by an upstream boundary condition. The hydraulic jump that occurs at the transition between super and sub-critical is not located at the location of the change in slope, but depends on boundary conditions. The critical depth d_c is denoted by d_g.*

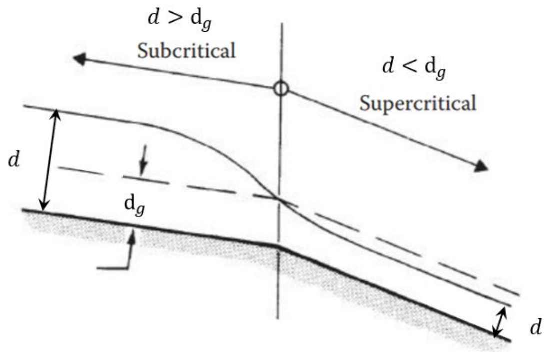


Figure 7.7: *The sub-critical flow (on the left) is determined by a downstream boundary condition. The transition from sub-critical to super-critical is located at location of the change in slope (the control section). The critical depth d_c is denoted by d_g.*

7.7.4 Analytical solution of the Bélanger equation (1)

To find an analytical solution, we have to make one restrictive assumption: the water-surface profile is only slightly different from its equilibrium profile d_e. Making this assumption enables us to write

$$\mathrm{d} = \mathrm{d}_e + \Delta\mathrm{d}, \tag{7.67}$$

where $\Delta\mathrm{d}$ is a very small quantity (relative to d_e) . Therefore,

$$\left(\mathrm{d}^3 - \mathrm{d}_e^3\right) = \left((\mathrm{d}_e + \Delta\mathrm{d})^3 - \mathrm{d}_e^3\right) = \left(\mathrm{d}_e^3\left(1 + \frac{\Delta\mathrm{d}}{\mathrm{d}_e}\right)^3 - \mathrm{d}_e^3\right). \tag{7.68}$$

Using the mathematical approximation $(1+x)^n \simeq 1 + nx$ when x is small, we obtain

$$\left(\mathrm{d}^3 - \mathrm{d}_e^3\right) \simeq \left(\mathrm{d}_e^3\left(1 + 3\frac{\Delta\mathrm{d}}{\mathrm{d}_e}\right) - \mathrm{d}_e^3\right) = 3\mathrm{d}_e^2\Delta\mathrm{d}. \tag{7.69}$$

We also have

$$\left(\mathrm{d}^3 - \mathrm{d}_c^3\right) \simeq \left(\mathrm{d}_e^3 - \mathrm{d}_c^3\right), \tag{7.70}$$

and the Bélanger equation reduces to

$$\frac{d\mathrm{d}}{dx} = \frac{d\left(\mathrm{d}_e + \Delta\mathrm{d}\right)}{dx} = \frac{d\left(\Delta\mathrm{d}\right)}{dx} = \frac{3i_b\mathrm{d}_e^2\Delta\mathrm{d}}{\mathrm{d}_e^3 - \mathrm{d}_c^3} \tag{7.71}$$

We can write

$$\frac{d\left(\Delta\mathrm{d}\right)}{dx} = \frac{\Delta\mathrm{d}}{\mathrm{L}}, \tag{7.72}$$

where L is a characteristic length given by

$$L = \frac{\mathrm{d}_e^3 - \mathrm{d}_c^3}{3i_b\mathrm{d}_e^2}. \tag{7.73}$$

Integrating the equation yields

$$\Delta\mathrm{d} = \Delta\mathrm{d}_0 \exp\left(\frac{x - x_0}{L}\right). \tag{7.74}$$

This equation tells us that, if at $x = x_0$ there is a water-surface elevation of $\Delta\mathrm{d}_0$ compared to the equilibrium profile d_e, then, assuming that the change in profile is very small, there will be a significant change in water depth over distances such that $|x - x_0| \gg L$.

7.7.5 Analytical solution of the Bélanger equation (2)

A particular case of the Bélanger equation occurs when $i_b = 0$. In that case, one should start from the equation:

$$\frac{d\mathrm{d}}{dx} = i_w = \frac{i_b - C_f U^2/(g\mathrm{d})}{1 - U^2/(g\mathrm{d})} \tag{7.75}$$

which reduces to

$$\frac{d\mathrm{d}}{dx} = -C_f \frac{\mathrm{d}_c^3}{\mathrm{d}^3 - \mathrm{d}_c^3}. \tag{7.76}$$

This equation can be integrated to give

$$\frac{\mathrm{d}^4}{4} - \mathrm{d}_c^3\mathrm{d} + C_f\mathrm{d}_c^3 x = \text{constant}. \tag{7.77}$$

7.8 Summary

After studying this chapter you should be able to:

1. Define friction slope i_f, bottom (wall) slope i_b and water surface slope i_w
2. Calculate the friction slope in a pipe
3. Give the general definitions of the hydraulic radius and the wetted perimeter
4. Calculate the hydraulic radius in pipe flow and associated hydraulic conductivity
5. Define and apply the Chézy equation
6. Apply the logarithmic profile for the flow to estimate the friction coefficient C_f and associated volume flux

7. Define the Bélanger equation (backwater equation)
8. Define the equilibrium depth (normal depth)
9. Know the different types of water-surface profiles
10. Know how to solve the Bélanger equation analytically

The Dutch corner

English	Nederlands
friction slope	wrijvingsverhang
backwater curve	verhanglijn (stuwkromme)