Chapter 2
Basic Components

Abstract We learned in Chap. 1 that blades of axial machines have profiles resembling aircraft wing profiles (aerofoils). In machines with large spacing between blades, this resemblance is strong, as with the hydraulic turbines and pumps studied up to now. With axial compressors, gas and steam turbines, blades are positioned closer together. The blades also cause a larger flow turning. A circumferential section results in a row of blade profiles with tangential spacing comparable to, or smaller than, the largest profile dimension. We then apply the term blade row or blade cascade. Radial machine rotor blades principally do not function as lifting objects. The blades constitute channels. The blade profiles have no resemblance to aerofoils. Channel flows may be accelerating (turbines) or decelerating (pumps, fans, compressors). With decelerating or diffusion flows, avoidance of separation between flow and geometry is difficult. As we have already learned, diffusers also occur as stator components of turbines. Aerofoils, cascades, channels and diffusers constitute the basic components of turbomachines, which we study in this chapter.

2.1 Aerofoils

2.1.1 Force Generation

Figure 2.1 sketches the streamlines close to an aerofoil in an oncoming flow with uniform velocity \( v_\infty \) and fluid density \( \rho_\infty \) far upstream. The streamlines are intended to curve due to camber of the profile (curvature in the longitudinal direction) and due to incidence of the flow (angle difference between the oncoming flow and the profile). To grasp the effect of the curvature, the simplest is considering an observer moving along with a fluid particle. This observer then stands still relative to the flow. For the relative coordinate system with origin on the observer, with x-axis along the flow and y-axis perpendicular to it, a centrifugal force must then be introduced with a value \( v^2/R \), perpendicular to the streamline, so along the y-axis, away from the centre of curvature (\( R \) is the radius of curvature). In the flow normal direction (y-axis) the momentum equals zero. Conservation of momentum thus means a balance of forces. This means that the centrifugal force must be kept in balance by a
pressure difference. The arrows perpendicular to the streamlines in Fig. 2.1 indicate the centrifugal force affecting the fluid particles. The pressure difference generated is indicated with + and − symbols. The consequence is that the pressure is lower than the distant pressure on the top side of the aerofoil, while on the bottom side it is higher than the distant pressure. It is said that a suction side and a pressure side form. The pressure difference generates a force in the upward direction, approximately perpendicular to the oncoming flow.

In fluid mechanics, it is proved that, with a lossless flow, the force on an aerofoil stands exactly perpendicular to the oncoming flow (Kutta-Joukowski theorem). In a flow with losses there is also a component in the flow direction. The perpendicular component is termed lift $L$. The component in the flow direction is termed drag $D$. These forces are usually expressed per span unit of the aerofoil (N/m). From fundamental fluid mechanics we recall that forces exerted by a flow onto objects (N), according to similitude theory, are proportional to $\rho_\infty v_\infty^2/2$ (= dynamic pressure, the difference between total pressure and static pressure; see Sect. 2.1.3) and to a characteristic object surface. We thus define lift coefficient and drag coefficient by

$$C_L = \frac{L}{\frac{1}{2}\rho_\infty v_\infty^2 c}, \quad C_D = \frac{D}{\frac{1}{2}\rho_\infty v_\infty^2 c}.$$ 

The symbol $c$ denotes the chord length of the aerofoil, which is a measure for the largest aerofoil dimension. It approximately equals the largest distance between two points at the aerofoil surface (defined below).

An aerofoil shape (Fig. 2.2) is characterised by a camber line and a thickness distribution. The camber line is a longitudinal line, such that points on the aerofoil surface are at half the thickness set perpendicularly to both sides of it. The camber line is approximately obtained by connecting the midpoints of the inscribed circles within the aerofoil. The leading side (A) is rounded with most aerofoil shapes, causing the inscribed circle to have a finite radius there. In theoretical aerofoil represen-
tations the trailing side (B) mostly has no thickness (a practical aerofoil is obtained by truncating). The *leading edge* is the final point of the camber line at the leading side and the *trailing edge* is the final point at the trailing side. The *chord* is the line segment between the leading and the trailing edges (AB).

The camber line is described by the distance to the chord (y) as a function of a coordinate (x) along the chord beginning at the leading edge and running to the trailing edge. The local thickness is commonly given as a function of the same coordinate. The angle between the chord and the oncoming flow is termed *chord angle* or *angle of attack*. Generally, chord angle means the angle between the chord and a geometric reference line while angle of attack means the angle between the chord and the oncoming flow direction. The concepts coincide with a free-standing aerofoil, but become different for an aerofoil in a machine setting. Most aerofoils have camber. As a consequence, lift is already generated with a zero value of the angle of attack. The angle of attack must then be negative in order to obtain zero lift. The position of the zero-lift defines a direction termed *zero-lift line*. The angle of attack may also be defined as the angle between the zero-lift line and the oncoming flow. The latter definition has the advantage of a zero lift with a zero value of the angle of attack. However, the zero-lift line cannot be indicated on the aerofoil a priori. From fluid mechanics it follows that this line is found with a good approximation by connecting the trailing edge to the point of maximum camber on the camber line.

### 2.1.2 Performance Parameters

Figure 2.3 sketches, as an illustration, the lift coefficient as a function of the angle of attack for a NACA 4412 aerofoil [1]. This aerofoil was formerly (before the contemporary common use of computational fluid dynamics techniques) typically applied in axial pumps and axial hydraulic turbines. The aerofoil shape is sketched in the figure as well. The NACA denomination refers to the classification by the National Advisory Committee for Aeronautics, the forerunner of the NASA, National Aeronautics and Space Administration. Also shown in Fig. 2.3 is the drag coefficient as a function of the lift coefficient. The tangent from the origin to the curve determines the angle of attack with maximum $L/D$ ratio. Use of the aerofoil with this angle of attack realises maximum efficiency in most applications. The corresponding lift coefficient is about 1.00, the drag coefficient about 0.01. The lift
increases with the angle of attack until an angle where the boundary layer at the suction side in the vicinity of the trailing edge separates from the surface. Beyond this angle, the lift coefficient decreases strongly and the drag coefficient increases strongly. This phenomenon is called *stall*, which means loss of proper functioning. An analogous stall phenomenon is observed with negative angles of attack, but aerofoils commonly are not designed to function well with highly negative angles of attack.

Boundary layer separation will be analysed in a following section, but one can immediately get an intuitive image of if for a very large chord angle, in particular when the profile is positioned perpendicularly to the flow. A large recirculation zone is then created downstream of the aerofoil. At the upstream side of the aerofoil, the pressure is higher than in the oncoming flow due to the flow retardation. Within the recirculation zone the pressure approximates the static pressure of the oncoming flow. The contribution to the drag by the pressure difference is termed *pressure drag*. With a weaker separation, as sketched in Fig. 2.4, pressure drag is generated.

**Fig. 2.3** Lift and drag coefficient for NACA 4412 at $Re = \frac{v_{\infty} c}{\nu} = 10^6$

**Fig. 2.4** Separated flow over an aerofoil
2.1 Aerfoils

This adds to the friction drag. The pressure drag is the resultant in the flow direction of the pressure on the surface. The friction drag is the resultant of the shear stress on the surface. Pressure drag is rather weak with attached flow, but separation causes a strong increase. Moreover, the curvature of the streamlines at the suction side is then strongly reduced. The consequence is that the pressure minimum at the suction side is weaker than with attached flow. This explains the lift decrease after separation.

2.1.3 Pressure Distribution

The pressure distribution for optimal operation of a NACA 4412 is sketched in Fig. 2.5. At the leading edge, the velocity is near to zero and the pressure is near to the stagnation pressure of the oncoming flow. The stagnation pressure or total pressure is the pressure obtained by bringing the flow to zero velocity in an adiabatic reversible way. This means that kinetic energy is converted into pressure energy. For constant density, the stagnation pressure is $p_o = p_\infty + \frac{1}{2} \rho_\infty v_\infty^2$. The pressure is somewhat lower in the leading edge zone. The trailing edge pressure is typically slightly higher than the pressure of the oncoming flow. The pressure coefficient shown in Fig. 2.5 is defined by $C_p = (p - p_\infty) / \frac{1}{2} \rho_\infty v_\infty^2$. It is common practice to plot this coefficient with negative values to the upside in order to have the suction side at the upside of the figure. At the pressure side of the aerofoil, the pressure is uniformly decreasing. The boundary layer flow is accelerating everywhere. At the suction side, flow accelerates at the leading edge, causing a pressure drop. Already with a low aerofoil load (small angle of attack), the minimum pressure is lower than the pressure of the oncoming flow (see Fig. 2.1). From the minimum pressure point to the trailing edge, the boundary layer at the suction side is subjected to an adverse pressure gradient.
2.1.4 Boundary Layer Separation

We can obtain more detailed understanding of the flow around an aerofoil by deriving the momentum equations for an infinitesimal streamtube close to a wall. Figure 2.6 sketches such a streamtube for the suction side. We consider a flow that follows the surface, in other words, that does not separate.

The axis in flow direction is denoted by \( x \). This direction approximately follows the surface. The normal direction is denoted by \( y \). Shear stress on a face of the control volume is denoted by \( \tau \). The flow between the streamtube considered and the wall has a braking effect on the streamtube. The flow between the streamtube and the free flow has a driving effect. The momentum theorem in the flow direction for an infinitesimal part of the streamtube, on condition of small curvature, is

\[
\rho v (dy) dv = -(dy) dp - \rho (dy) dU + (dx) d\tau.
\]

From this:

\[
\rho v \frac{dv}{dx} = -\frac{dp}{dx} - \rho \frac{dU}{dx} + \frac{d\tau}{dy}.
\]

(2.1)

From now on, we assume constant density, but extension for variable density is possible. With constant density, the effect of gravity onto the pressure may be expressed by considering pressure as relative to the hydrostatic pressure. This is relevant for liquid flow. The effect of gravity may simply be ignored for gas flow. We thus simplify (2.1) to

\[
\rho v \frac{dv}{dx} = -\frac{dp}{dx} + \frac{d\tau}{dy}.
\]

(2.2)

Fig. 2.6 Infinitesimal streamtube close to a wall and velocity profile evolution with an adverse pressure gradient
We first consider a flow with a zero pressure gradient. Far from the wall, the shear stress is zero. There is a certain value on the wall. So $dt/dy < 0$. The momentum equation demonstrates that friction reduces the velocity within the streamtube in the flow sense. The height of the streamtube thus increases in the flow sense. The same applies to the entire zone close to the wall, where flow is retarded by the wall friction. The zone affected by the wall friction is called the *boundary layer*. Its thickness is denoted by $\delta$ in Fig. 2.6. Note that this thickness cannot be determined precisely. The shear stress is partly of molecular nature due to the relation

$$\tau = \mu \frac{dv}{dy},$$

(2.3)

where $\mu$ is the dynamic viscosity coefficient. We recall from fluid mechanics that friction is generated between two adjacent fluid layers with different velocity by the chaotic motion of the microscopic fluid particles (molecules or atoms) around the average macroscopic motion. Fluid particles leap from the fast layer to the slower one, where they arrive with a higher average momentum and so push forward the slower layer. Oppositely, fluid particles leaping from the slower layer to the faster one produce a breaking effect. The resulting effect is described macroscopically by Eq. (2.3), with the viscosity coefficient $\mu$ as a positive quantity. According to the momentum Eq. (2.2), friction affects the velocity change with $\mu/\rho$. We thus define the kinematic viscosity coefficient

$$\nu = \frac{\mu}{\rho}.$$  

(2.4)

The dimension of kinematic viscosity is $m^2/s$. The value for water at atmospheric temperature is about $10^{-6}$ $m^2/s$. The value for air at atmospheric temperature and pressure is about $15 \times 10^{-6}$ $m^2/s$. So water and air are not highly viscous fluids. This results in a very thin boundary layer. It is demonstrated in fluid mechanics that, on a smooth wall without a pressure gradient, boundary layer thickness is approximately

$$\delta / x = 5 / \sqrt{\frac{x v_\infty}{\nu}}.$$ 

Here, $v_\infty$ represents the velocity far away from the wall (outside the boundary layer zone) and $x$ the distance covered by the boundary layer. A boundary layer thickness (with laminar flow) maximally amounts to some thousandths of the covered length. It is very important to keep in mind the small thickness of a boundary layer in boundary layer analyses. When drawing a boundary layer, the thickness must
necessarily be exaggerated, which creates a false impression. Turbulent flow contains eddies (whirling flow patterns) as a result of the fragmentation due to stretching and bending of the vortices generated by shear zones. Eddies have a macroscopic size. Their motion is strongly chaotic around an average flow, similar to, but on a far larger scale, than the microscopic fluid particles. The turbulent motion thus also generates shear stress on the average flow. This stress is expressed by an eddy viscosity $\mu_t$ according to

$$\tau = (\mu + \mu_t) \frac{dv}{dy}.$$  \hspace{1cm} (2.5)

The eddy viscosity coefficient $\mu_t$ is no fluid characteristic and depends on the local flow. It is important to realise that $\mu_t$ is a positive quantity and that it may be up to 100 times bigger than the molecular viscosity coefficient. So, due to the turbulence, the boundary layer thickness increases significantly. The thickness stays small however compared to the covered length, namely some hundredths. From the positive values of $\mu$ and $\mu_t$ and the negative value of $d\tau/dy$, we understand that the velocity within the streamtube decreases in flow sense, due to friction, but that inversion of the flow sense is impossible. Separation thus cannot be caused by the viscosity effect only.

In order to understand the role of the pressure gradient, we must also analyse the pressure variation in the normal direction. The easiest way is with a relative frame attached to a fluid particle, as used in Sect. 2.1.1, leading to a balance between the normal pressure gradient and the centrifugal force caused by the curvature of the streamline:

$$\frac{1}{\rho} \frac{dp}{dy} = \frac{v^2}{R}.$$  \hspace{1cm} (2.6)

$R$ is the radius of curvature. It is assumed in Eq. (2.6) that friction forces do not contribute. As is clear in Fig. 2.6, there is also shear stress on the inlet and outlet faces of the infinitesimal streamtube. A stress change in the flow direction thus contributes to the force in the normal direction. As boundary layers are very thin, with changes in the normal direction being much bigger than in the flow direction, the contribution of friction may be ignored.

A consequence of Eq. (2.6) is that the pressure distribution over an aerofoil, as sketched in Fig. 2.5, is only slightly dependent on the fluid viscosity, except maybe with very viscous fluids. Another consequence of Eq. (2.6) is an almost identical pressure variation over the various streamlines in a boundary layer, as the radius of curvature $R$ is very large compared to the boundary layer thickness. The decrease of momentum, arising on the various streamlines according to Eq. (2.2) with an adverse pressure gradient, is thus about the same for all streamlines. Consequently, the velocity decrease is relatively stronger as the streamline is nearer to the surface.
This causes, as sketched in Fig. 2.6, the velocity profile to become more concave as the flow evolves. With a sufficiently large adverse pressure gradient, the velocity close to the profile surface may attain a very low value, causing separation downstream of this position. With the reasoning with streamtubes following the surface, as sketched in Fig. 2.6, it cannot be analysed how the transition from attached flow to separated flow exactly occurs, as the velocity within a streamtube cannot change sign. The reasoning however explains the origin of separation, which is sufficient here. We conclude that separation may occur in the trailing edge zone at the suction side, if the adverse pressure gradient is sufficiently large. We also conclude that separation cannot occur at the pressure side if the flow is everywhere accelerating. We further notice that a turbulent boundary layer has a far better resistance to separation than a laminar one. A turbulent boundary layer is much thicker than a laminar one, but turbulence generation is most intense close to the wall. With a turbulent boundary layer, the near-wall velocity gradient is much higher and thus more momentum is present near the wall than within a laminar boundary layer.

2.1.5 Loss Mechanism Associated to Friction: Energy Dissipation

The loss mechanism associated to friction in a boundary layer may be understood by analysing the work by friction exerted on the streamtube shown in Fig. 2.6. The bottom streamline is a stationary wall (velocity equal to zero). The overlying flow exerts positive work \((\tau + d\tau)(v + dv)\) and drives the streamtube. The underlying flow brakes by the negative work \((-\tau v)\). The net work leads to the energy equation, in the absence of heat transfer:

\[
\rho (dy)v d(h + \frac{1}{2}v^2) = (\tau + d\tau)(v + dv)dx - \tau vdx,
\]

or

\[
\rho v \frac{dh}{dx} = \frac{d}{dy}(\tau v).
\]  \tag{2.7}

Potential energy, if relevant, is included in the enthalpy. The shear stress \(\tau\) varies in the y-direction from the wall value to zero in the main flow. Velocity varies from zero to the main flow value. The \(\tau v\) quantity thus goes from zero to zero through a positive maximum. Thus, friction work causes a redistribution of total enthalpy within the boundary layer, but, with an adiabatic wall, the total enthalpy flux stays constant within the entire boundary layer. This follows from the integration of Eq. (2.7) in the y-direction.
The work equation associated to the momentum Eq. (2.2) is

\[ \rho v \frac{d}{dx} \left( \frac{1}{2} v^2 \right) = -v \frac{dp}{dx} + v \frac{d\tau}{dy}. \]

or

\[ \rho v \left( \frac{d}{dx} \left( \frac{1}{2} v^2 \right) + \frac{1}{\rho} \frac{dp}{dx} \right) = v \frac{d\tau}{dy}. \] (2.8)

From now on, we assume again constant density. The work Eq. (2.8) then means that the mechanical energy \( E_m = \frac{1}{2} v^2 + \frac{1}{\rho} p \) decreases due to the displacement work of the friction force, since \( d\tau/dy < 0 \).

Subtracting the work equation from the energy equation results in

\[ \rho v \frac{de}{dx} = \tau \frac{dv}{dy}. \] (2.9)

The part of the work \( \tau \frac{dv}{dy} \) is the deformation work: total work minus displacement work. This work is always positive in view of (Eq. 2.5). Friction thus results in increase of internal energy (Eq. 2.9) and decrease of mechanical energy (Eq. 2.8) with constancy of the flux of the total energy (Eq. 2.7): mechanical energy plus internal energy. The conversion of mechanical energy into internal energy is called energy dissipation.

With the boundary layer of Fig. 2.6 there is no exchange of work and heat with the surroundings. Therefore, the flux of total energy is constant and the decrease of mechanical energy is exactly equal to the increase of internal energy. So, energy dissipation may be seen in both ways. This is not a general result, however. Strictly, energy dissipation is caused by the deformation work of the friction. This may be understood by adding work done by an external force on the boundary layer in Fig. 2.6. The same work term then adds to the balance of total energy and the balance of mechanical energy. In the difference of both balances, the work term cancels. This demonstrates that actually the balance of internal energy (Eq. 2.9) determines the dissipation. This observation may cause confusion since in Chap. 1 the displacement work of the friction force resulting in mechanical energy decrease was immediately denoted as dissipation. Strictly, this is incorrect, as the deformation work is the dissipative part of the friction work. The result is correct in value, because the magnitude of the integral across the boundary layer thickness of the displacement work equals the magnitude of the integral of the deformation work, since the integral of the total work equals zero with a shear flow on a stationary wall. It is essential for this result that the friction acts on a stationary wall.

From the analysis with the differential equations it follows as well that the energy dissipation does not exactly equal the product of the shear stress at the wall and
the average velocity in a channel, as obtained in a one-dimensional analysis. This product has the correct order of magnitude, but the exact result depends on the distribution of the shear stress and the velocity gradient across a channel section. With a trapezoidal velocity profile and with an infinitesimally thin transition from core to wall, it follows that the energy dissipation integral (Eq. 2.9) equals the product of the velocity in the flow core and the shear stress at the wall.

The reasoning based on the energy Eq. (2.7) and the work Eq. (2.8) demonstrates that the wall friction affects the entire boundary layer. Energy dissipation by friction is thus no local phenomenon. Moreover, the dissipation continues downstream of the profile. Figure 2.7 illustrates how boundary layers at the suction and the pressure sides merge at the trailing edge of an aerofoil and how a wake is generated in which further energy dissipation occurs.

As mentioned above, the drag of the aerofoil is resolved into friction drag and pressure drag. Energy dissipation associated to drag may be resolved in two parts as well, termed friction loss within the boundary layers and mixing loss within the wake downstream of the aerofoil. The term mixing loss expresses the energy dissipation by the mixing of the flows from the suction side and the pressure side with each other and with the surrounding flow. Figure 2.7 makes clear that the expansion of the wake, in principle, continues up to an infinite distance from the aerofoil. So, it is very difficult to estimate the total loss by an integral of the energy dissipation within the fluid. The result of the integration is known a priori however. Let us consider, instead of a stationary aerofoil in a flow with oncoming velocity \( v_\infty \), the motion of an aerofoil with velocity \( v_\infty \) in a stationary atmosphere. For the last case, the totally dissipated energy per time unit is \( \bar{D} \bar{v}_\infty \), as all work is dissipated. Obviously, the amount must be the same for the stationary aerofoil in the steady flow. The following must thus apply (where S is a surface enclosing the aerofoil at a very large distance):

\[
\bar{D} \bar{v}_\infty = \int_S (h_0 - E_m) \rho \bar{v} \bar{n} \, dS = -\int_S E_m \rho \bar{v} \bar{n} \, dS. \tag{2.10}
\]

\(Fig. 2.7\) Merging boundary layers at a trailing edge with generation of a wake.
2.1.6 Profile Shapes

The optimal profile shape strongly depends on the particular application. The NACA 4412 shape used in Fig. 2.3, is, as already mentioned, rather suitable for applications in pumps and hydraulic turbines. This profile was used frequently in older designs. The intended flow deflection is small, the Reynolds number moderately high to high, i.e. in the order of 1 to 5 $10^6$ and with a high degree of turbulence within the main flow. There first is flow acceleration at the suction side, which keeps the boundary layer laminar (the laminar boundary layer becomes more stable in accelerating flow). Deceleration follows after that. With a high Reynolds number and high turbulence in the main flow, the boundary layer quickly becomes turbulent when subjected to an adverse pressure gradient (the laminar boundary layer becomes less stable in decelerating flow). This is intended, as a turbulent boundary layer better resists separation than a laminar one. An aerofoil shape with a pressure distribution as sketched in Fig. 2.5 is therefore termed a turbulent profile, which means that the boundary layer at the suction side is turbulent on a large part of it. The profile shape is not optimal for applications in pumps and hydraulic turbines. The predominantly turbulent boundary layer at the suction side allows a high adverse pressure gradient. Consequently, the attainable lift coefficient is high. But the drag coefficient is rather high as well, due to the higher friction than with laminar flow. Another disadvantage is the strong pressure minimum at the suction side, which may cause cavitation (local water evaporation) with some applications. Modern profile shapes have a less deep suction pressure minimum. The laminar part of the leading edge boundary layer is larger, generating a lower drag coefficient. Nowadays, these profiles are designed with computational fluid dynamics techniques.

Figure 2.8 shows an aerofoil and the accompanying pressure distribution for the middle part of a wind turbine blade [5]. The pressure distribution is completely different from the NACA 4412. Flow circumstances are here: low deflection, slight acceleration within the general flow, rather high Reynolds number, i.e. 3–5 $10^6$ and low turbulence in the main flow. Strong acceleration occurs at the leading edge part of the suction side, followed by a zone of weak acceleration. The boundary layer stays laminar until the end of the weak acceleration zone. This succeeds due to the low turbulence in the oncoming flow. From the beginning of the deceleration phase, the boundary layer becomes turbulent here as well. This succeeds due to the rather high Reynolds number. At first, acceleration occurs at the pressure side due to the high aerofoil thickness. Then follows a weak deceleration intended to increase the pressure difference between the pressure and the suction sides again. The obtainable lift coefficient is moderately high ($\sim 1.00$). The drag coefficient is low ($\sim 0.0075$), as the boundary layer at the suction side is laminar over a rather significant part. The aerofoil shape is termed a laminar profile. The shape is similar to profiles applied in wings of aircraft.

Present-day aerofoil profiles use the features of the profile shape of Fig. 2.8: strong acceleration followed by weak acceleration at the suction side leading edge,
Aerofoils with a limitation of the minimum pressure; deceleration in the middle part of the pressure side in order to enhance the lift. Gradients within the zones and the extensions of the zones strongly vary from application to application. The optimum is critically determined by the Reynolds number, the mean pressure gradient in the flow (globally accelerating or decelerating flow) and the turbulence level in the core of the flow. There is no universal optimum. Determination of the optimum requires the application of advanced computational methods.

2.1.7 Blade Rows with Low Solidity

With the axial turbine or the axial pump discussed in Chap. 1, the blade profiles do not have the same performance as isolated blades, because the suction and pressure sides of neighbouring blades interact. This may cause both lift increase or lift decrease. The zero-lift direction changes and drag stays approximately the same. Figure 2.9 sketches a blade row for decelerating flow (pump, fan, compressor). The tangential distance between the blades is called spacing ($s$) or pitch. The ratio of the chord ($c$) to the spacing is termed solidity ($\sigma = c/s$). With blade systems, the term solidity generally refers to the ratio of the blade area to the flow area. An approximate blade area is typically applied, obtained by integration of the chord: $\int c dr$. With a good approximation, the zero-lift line is found by connecting the trailing edge to the point of maximum deflection on the camber line, taking the maximum

![Fig. 2.8 Aerofoil shape and pressure distribution for a central part of a wind turbine blade](image-url)
on a reduced chord between the trailing edge and a point obtained by projection in the axial direction of the trailing edge of the neighbouring profile [7].

Figure 2.10 demonstrates how the lift coefficient changes. The lift coefficient shows a nearly linear variation over a broad angle of attack range. This is the range with no boundary layer separation. The slope of the lift curve changes with a factor $k$, plot in the figure. Results originate from measurements on various profile shapes, but all of them with a relative thickness of 8% [7]. The angle in the diagram is between the zero-lift line of the isolated profile and the tangential direction. The diagram becomes unreliable for high solidity. It may be used confidently for $\sigma = c/s < 1.3$. At higher solidity it becomes difficult to derive the blade row characteristics from the characteristics of isolated blades (see next section).

### 2.2 Linear Cascades

#### 2.2.1 Relation with the Real Machine

Flow within the blade passages of an axial turbomachine is three-dimensional. A two-dimensional approximation is obtained by making a cylindrical section of the
blades at the mean radius and by unrolling it into a plane. A linear cascade or blade row is built by setting up prismatic blades with the profiles obtained. In principle, an infinite number of blades are required in order to keep the periodicity of the blades in the original machine. The two-dimensional flow is representative for the real three-dimensional flow. The expressions for axial momentum are the same in both configurations. Moment of momentum with the three-dimensional flow corresponds to tangential momentum with the cascade (tangential = circumferential). Constant tangential momentum along the blade height with the linear cascade is then equivalent to constant angular momentum \((v_u r = \text{constant})\) along the radius in the real machine. This means then that the three-dimensional flow is a so-called free vortex flow. The term means swirling flow with constant angular momentum. In the pioneering time, axial turbomachines were built with free vortex flow because of this theoretical correspondence. They show certain disadvantages discussed later (Chap. 13: axial compressors; Chap. 15: axial and radial turbines). Present-day machines are designed with some degree of forcing of the vortex \((v_u r \neq \text{constant})\) along the radius. The deviation to the free vortex distribution is not very significant, however. With free vortex blades in a cylindrical axial turbomachine, an exact transformation exists between the flow in the machine and the linear cascade (of course with top and bottom walls added). This does no longer apply with some forcing on the vortex. For instance, no cylindrical streamsurfaces exist anymore. The analogy between the three-dimensional flow within the machine and the two-dimensional flow within the linear cascade is only met approximately. Nevertheless, studying the characteristics of linear cascades is very instructive as the flow within a linear cascade is still representative for the flow in an average cylindrical section of a real machine.

### 2.2.2 Cascade Geometry

The cascade is determined by the shape of the blades, their position to the axial direction and their spacing (tangential distance between the blades). The term pitch is often used for spacing, but pitch may also mean, similarly to the term with screws, the axial distance covered by the flow when it makes a full turn of 360°. This applies in particular to propellers and wind turbines with adjustable rotor blades. Then often, also the term pitch angle is used. Results are also a function of the positioning of the blades relative to the flow. The profile shape is, as with aerofoils, determined by the camber line and the thickness distribution. The profile size is determined by the chord. Solidity \(\sigma\) means the ratio of the chord to the spacing. Some characteristics are shown in Fig. 2.11. The camber angle \(\phi\) is the angle difference between the tangents at the camber line on the leading and trailing edges. Profile orientation relative to the axial direction is determined by the chord angle or stagger angle \(\gamma\). The inlet velocity \(w_i\) forms an angle of attack \(\alpha\) with the chord, an angle \(\beta_i\) with the axial direction and an angle of incidence \(i\) with the tangent at the camber line on the leading edge. The angle of incidence is positive when the flow turning increases
by it. The outlet velocity $w_2$ forms with the inlet velocity $w_1$ an *angle of deflection* $\theta$, with the tangent at the camber line on the trailing edge an *angle of deviation* $\delta$ and with the axial direction an angle $\beta_2$. The angle of deviation is positive when it causes a decrease of $\theta$.

### 2.2.3 Flow in Lossless Cascades: Force Components

Figure 2.12 sketches the flow through a rotor cascade with decelerating flow. We apply the fundamental laws (for a constant density fluid) to a cascade with profiles on spacing $s$. The flow velocities in the relative frame $w$ have $w_a$ into the axial direction and $w_u$ into the tangential direction (= circumferential direction) as their components. Similarly, the force $L$ exerted on the blades has $L_a$ and $L_u$ as components. The velocities $w_1$ and $w_2$ are considered sufficiently far from the cascade, so that velocity is assumed to be constant in Sections 1 and 2.

We select a right-handed coordinate frame with the x-axis in the axial direction, positive in the through-flow sense, and the y-axis in the circumferential direction, positive in the running sense of the rotor. The machine must be left turning ($\Omega = -\Omega \hat{I}_x$) in order that the z-axis in the radial direction has the outward sense as positive sense. This is a minor complication. A right-handed coordinate frame with a right turning machine would be achieved by an x-axis in the running sense, a y-axis in the axial direction in the through-flow sense and a z-axis in the radial direction in the outward sense ($\Omega = \Omega \hat{I}_y$). With this last convention, angles are calculated with respect to the circumferential direction and vary from 0° to 180°.
This choice was very often made in the past. It has the disadvantage that application of the goniometric tangent function is difficult. Due to the general use of electronic calculation devices, the first convention (axial convention) is preferred at present, with angles varying from $-90^\circ$ to $+90^\circ$. This convention has already been applied systematically in Chap. 1. For the entire book, we exclusively reason on left turning machines. Both left and right turning machines actually occur in practice (some illustrations concern right turning machines). More generally, we take a coordinate frame with first axis in the meridional direction (this is the tangent to the intersection of the average circumferential streamsurface with a meridional plane), positive into the through-flow sense; second axis in the circumferential direction, positive in the running sense; third axis perpendicular to the average streamsurface, positive in the outward sense.

**Fig. 2.12** Force exerted onto a cascade blade (rotor pump)
Tangential velocity components are negative for the cascade in Fig. 2.12, with lift components $L_a$ and $L_u$ being negative as well. We reason in the relative frame. As centrifugal force and Coriolis force lie in the radial direction with axial machines, these forces do not intervene with momentum relations in the axial and tangential directions. The derived relations thus automatically apply to stator cascades, as we always apply the momentum laws in an algebraically consistent way. As a consequence, the derived relations also apply to cascades with accelerating flow (this will be verified later on). The pump rotor cascade in Fig. 2.12 is the most complex one to analyse ($w_u, L_a$ and $L_u$ being negative). That is the reason why we take it as an example.

The mass conservation law is:

$$w_{iu} s = w_{2u} s = w_a s.$$  

The work equation (relative system, no losses: Eq. (2.8) with $\tau=0$) is

$$p_{01r} - p_{02r} = 0 \quad \text{or} \quad p_1 + \frac{\rho}{2} w_i^2 = p_2 + \frac{\rho}{2} w_2^2,$$

from which

$$p_2 - p_1 = \frac{\rho}{2} (w_i^2 - w_2^2).$$

With $\Delta p = p_2 - p_1, q_1 = \frac{\rho w_i^2}{2}$ and $w_a = w_2 \cos \beta_2 = w_i \cos \beta_1$,

it follows that:

$$C_p = \frac{\Delta p}{q_1} = 1 - \frac{\cos^2 \beta_1}{\cos^2 \beta_2}. \quad (2.11)$$

The term $\Delta p/q_1$ is often denoted by $C_p$. A criterion for maximum cascade load with decelerating flow is: $C_p = 0.5 \ (w_2/w_i \approx 0.7)$. This results in a relation between $\beta_1$ and $\beta_2$. For instance, corresponding values are $\beta_1 = -60^\circ$ and $\beta_2 = -45^\circ$, $\beta_1 = -55^\circ$ and $\beta_2 = -35^\circ$.

In order to apply the momentum conservation law, we consider the dashed lined contour in Fig. 2.12. It forms a control volume with two periodic streamsurfaces and two planes parallel to the cascade front. On the front and back surfaces, pressures are $p_1$ and $p_2$. Pressure forces on the periodic streamsurfaces counterbalance. The force by the blade on the flow is $-L_u$.

The momentum balance in the tangential direction is

$$-L_u = \rho \ s \ w_a (w_{2u} - w_{1u}),$$

or

$$L_u = \rho \ s \ w_a (w_{1u} - w_{2u}). \quad (2.12)$$
The momentum balance in the axial direction is

\[-L_u + s\ p_1 - s\ p_2 = 0,\]

or

\[L_u = s(\ p_1 - p_2 ) = s\ \frac{\rho}{2}(w_{2u}^2 - w_{1u}^2).\]  \hspace{1cm} (2.13)

The circulation around the contour in the positive sense, as periodic parts intervene with counterbalancing amounts, is

\[\Gamma = \oint \vec{w}\ d\vec{l} = s(-w_{1u} + w_{2u}) = s(w_{2u} - w_{1u}).\]

Combination results in

\[L_u = -\rho \ \Gamma \ w_a \quad \text{and} \quad L_a = \frac{\rho \Gamma}{2}(w_{1u} + w_{2u}).\]  \hspace{1cm} (2.14)

The result from these formulae is that the force \(L\) is perpendicular to the velocity \(w_m\), which has \(w_a\) and \((w_{1u} + w_{2u})/2\) as its components and that the magnitude of \(L\) is

\[|L| = \rho \ \Gamma \ w_m.\]  \hspace{1cm} (2.15)

This relation is termed the Kutta-Joukowski law for cascades. It is entirely analogous to that for aerofoils, being \(|L| = \rho_\infty \ |\Gamma| \ w_\infty.\) The expression for aerofoils may be derived from Eq. (2.15), by keeping \(\Gamma\) and allowing \(s\) to increase to infinity. Then \(w_1 = w_2 = w_\infty\) becomes the velocity of the parallel flow.

### 2.2.4 Significance of Circulation

The work equation on a streamline through the cascade is Eq. (2.2), but with \(dt/dy=0\) in the core of the flow (negligible shear force), which we write for a rotor cascade as

\[\rho \ w \ \frac{dw}{dx} + \frac{dp}{dx} = 0.\]  \hspace{1cm} (2.16)

The force balance in the direction perpendicular to the streamline is similar to Eq. (2.6):

\[\frac{1}{\rho} \ \frac{dp}{dy} = \frac{w^2}{R}.\]  \hspace{1cm} (2.17)
Equation (2.16) may be integrated along a streamline into

\[
\frac{1}{2} w^2 + \frac{1}{\rho} p = \text{constant.} \tag{2.18}
\]

With the cosine rule on the velocity triangles follows

\[
w^2 = u^2 + v^2 - 2uv_u.
\]

Thus:

\[
\frac{1}{2} v^2 + \frac{1}{\rho} p + \frac{1}{2} u^2 - uv_u = \text{constant.} \tag{2.19}
\]

This means that, for constant mechanical energy of the incoming flow in the absolute frame and uniform \( v_u \), the constant in Eq. (2.19) and thus in Eq. (2.18) is the same on all streamlines. Eq. (2.18) then also implies

\[
w \frac{dw}{dy} + \frac{1}{\rho} \frac{dp}{dy} = 0. \tag{2.20}
\]

Combined with Eq. (2.17), this gives

\[
w \frac{dw}{dy} + \frac{w^2}{R} = 0. \tag{2.21}
\]

The significance of Eq. (2.21) may be understood by calculating the circulation on an infinitesimal contour consisting of two pieces of neighbouring streamlines connected with straight segments as shown in Fig. 2.13 (\( x \) is streamline direction, \( y \) is normal direction), and by applying Stokes’ circulation theorem, which is an integral theorem for the rotor of a vector quantity:

\[
\oint \bar{w} \cdot d\bar{l} = \int_{S} (\nabla \times \bar{w}) \cdot \bar{n} \, dS,
\]

where \( S \) is a surface spanned by the contour and \( \bar{n} \) is the unit normal vector on the surface in the sense corresponding with the sense on the contour. On the infinitesimal contour of Fig. 2.13, this gives, with \( R \) the radius of curvature of the
mean streamline and with $\omega$ the value of the rotor of the relative velocity vector in the z-direction:

$$
(w + \frac{1}{2} \frac{dw}{dy} (R + \frac{1}{2} dy) d\theta - (w - \frac{1}{2} \frac{dw}{dy} (R - \frac{1}{2} dy) d\theta = \omega Rd\theta dy,
$$

or

$$
\frac{dw}{dy} Rd\theta + w dy d\theta = \omega Rd\theta dy.
$$

Thus

$$
\frac{dw}{dy} + \frac{w}{R} = \omega.
$$

(2.22)

With (Eq. 2.22), we understand that (Eq. 2.21) means that the rotor of the velocity vector is zero everywhere in the core of the flow. Such a flow is called \textit{irrotational}. The meaning of the term is that the rotational speed of each fluid particle is zero, as the rotational speed is equal to half of the rotor of the velocity vector (see fluid mechanics). The consequence is that circulation is zero on every contour non-enclosing the blade and that the circulation on a contour enclosing the blade is the same for every contour around the blade. Strictly, this result is only valid for lossless flow, as the mechanical energy has to be the same for every streamline. In flow with losses, the result remains approximately valid for every contour around the blade as long as this contour does not enter the boundary layers on the blades. The contour has to cut the wake, of course, which causes a small error on the strict equality of the circulation on each contour around the blade.

\subsection{2.2.5 Flow in Lossless Cascades: Work}

When the cascade moves into the tangential direction at a velocity $u$, the absolute velocities $v$ may be derived from $u$ and $w$ by constructing the velocity triangles:

$$
v_{1u} = u + w_{1u}; \quad v_{2u} = u + w_{2u}; \quad v_a = w_a.
$$
Formula (2.12) then becomes:

\[ L_u = \rho \, s \, v_a (v_{1u} - v_{2u}). \]

The power transferred to the fluid becomes

\[ P = -L_u u = \rho \, s \, v_a (v_{2u} - v_{1u}) u. \]

The power per mass flow rate unit, i.e. the work per mass unit, becomes

\[ \Delta W = \frac{-L_u u}{\rho \, s \, v_a} = u(v_{2u} - v_{1u}). \]

This is Euler’s formula, applied to an axial machine \((u_1 = u_2)\).

It is informative to remark that the work also may be found from \(-\tilde{L}\tilde{v}_m = -\tilde{L}_1 - \tilde{L}_m = -\tilde{L}_2\). So, the work done on the flow by an active force, indeed, is the force multiplied with the displacement, as applied in Chap. 1. This seems to be evident at a first glance. But that is not the case, as we analyse below, for a flow with losses.

### 2.2.6 Flow in Cascades with Loss: Force Components

Loss results in a total pressure drop (Eq. 2.8), which we may express by

\[ p_{01r} - p_{02r} = (p_1 + \frac{\rho \, w_1^2}{2}) - (p_2 + \frac{\rho \, w_2^2}{2}), \]

being a positive quantity. For further analysis, we assume equal loss on all streamlines. A loss coefficient, based on inlet dynamic pressure, is then

\[ \xi = \frac{p_{01r} - p_{02r}}{q_1}. \]

With \( F \) the force of the flow onto the profile, then

\[ F_u = \rho \, s \, w_a (w_{1u} - w_{2u}) = \rho \, s \, w_a^2 (\tan \beta_1 - \tan \beta_2), \]
\[ F_a = s \, (p_1 - p_2) = \frac{s \, w_a^2}{2} (\tan \beta_2 - \tan \beta_1) + s \, (p_{01r} - p_{02r}) \]
\[ = \rho \, s \, w_a^2 \, \tan \beta_m (\tan \beta_2 - \tan \beta_1) + s \, (p_{01r} - p_{02r}), \]
with \( \tan \beta_m = \frac{\tan \beta_1 + \tan \beta_2}{2} \).

The drag follows from (see Fig. 2.12):

\[
D = (F_{u1}\tilde{I}_u + F_{a1}\tilde{I}_a)(\sin\beta_m\tilde{I}_u + \cos\beta_m\tilde{I}_a) \\
= F_u\sin\beta_m + F_a\cos\beta_m = s(p_{01r} - p_{02r})\cos\beta_m.
\] (2.23)

The drag components are

\[
D_a = s(p_{01r} - p_{02r})\cos^2\beta_m, \\
D_u = s(p_{01r} - p_{02r})\sin\beta_m\cos\beta_m.
\]

The lift components are

\[
L_u = \rho s w_a^2 (\tan\beta_1 - \tan\beta_2) - s(p_{01r} - p_{02r})\sin\beta_m\cos\beta_m, \\
L_a = -\rho s w_a^2 \tan\beta_m (\tan\beta_1 - \tan\beta_2) + s(p_{01r} - p_{02r})\sin^2\beta_m.
\]

The magnitude of the lift for the cascade in Fig. 2.12 \((\tan\beta_1 - \tan\beta_2 < 0)\) is

\[
L = \rho s w_a^2 \frac{\cos^2\beta_m}{\cos\beta_m} (\tan\beta_2 - \tan\beta_1) + s(p_{01r} - p_{02r})\sin\beta_m.
\]

From this follows:

\[
C_{L1} = \frac{L}{q_1c} = \frac{2}{\sigma} \frac{\cos^2\beta_m}{\cos\beta_m} (\tan\beta_2 - \tan\beta_1) + \frac{\xi_1}{\sigma}\sin\beta_m,
\]

\[
C_{D1} = \frac{D}{q_1c} = \frac{\xi_1}{\sigma}\cos\beta_m.
\]

Elimination of \( \xi_1 \) results in

\[
C_{L1} = \frac{2}{\sigma} \frac{\cos^2\beta_1}{\cos\beta_m} (\tan\beta_2 - \tan\beta_1) + C_{D1}\tan\beta_m.
\]
This relation implies that the lift coefficient does not depend solely on the cascade solidity and the flow angles, but on the drag coefficient as well.

With the above formulae it should be taken into account that the $\beta$-angles are negative with the cascade considered in Fig. 2.12. This means that lift decreases due to losses. When deriving the relations, sign conventions have been applied consistently. The expressions for $F_u$ and $F_a$ have general validity, also for a turbine cascade ($\tan \beta_1, \tan \beta_2 > 0$, $F_u > 0$, $F_a > 0$). The expression for drag remains the same with a turbine. The expression for lift, with lift being a positive quantity, becomes ($\beta_m < 0$) with a turbine:

$$C_{L1} = \frac{2 \cos^2 \beta_1 \cos \beta_m}{\tan \beta_1 - \tan \beta_2 \sin \beta_m}.$$

It is remarkable that, with a given deflection in a turbine, losses cause lift increase. The different behaviour compared to the pump can be understood by analysing the four configurations shown in Fig. 2.14. The four possible combinations of the sign of the circulation and the mean tangential velocity occur. First, the correspondence of the sign combinations with Eqs. (2.14) may be verified. The tangential components of lift and drag have the same sense with pump cascades. For a given change of tangential velocity, this means given $F_u$, $D_u$ decreases the magnitude of $L_u$. This is just the opposite with a turbine cascade.

### 2.2.7 Flow in Cascades with Loss: Energy Dissipation and Work by Drag Force

Equation (2.23) for drag implies

$$Dw_m = \rho s w_m \cos \beta_m \frac{P_{01r} - P_{02r}}{\rho} = \rho s w_a \frac{P_{01r} - P_{02r}}{\rho},$$

or

$$Dw_m = m q_{IRR}.$$

Equation (2.24) is similar to the formerly found expression for energy dissipation with an aerofoil Eq. (2.10). The right-hand part in Eq. (2.24) represents the difference between the mechanical energy fluxes at the inlet and outlet planes of the cascade. The term $Dw_m$ is thus the total amount of energy dissipation. This result expresses that, with flow over a stationary cascade (we reason in the relative frame), energy dissipation equals the displacement work of the drag force exerted onto the average flow.

The finding that, for a given flow deflection, drag affects lift, is due to the contribution of the drag to the tangential velocity change. For a moving
cascade, this means that drag contributes to the work. The work by the drag force on the fluid is \(-D\vec{u}\). Analogously to the lift, we can verify the meaning of \(-mDv\).

It follows that

\[-D\vec{u} = -D\vec{v}_m + D\vec{w}_m,\]

with

\[-D\vec{u} = \rho sw_a \Delta W \quad \text{and} \quad D\vec{v}_m = \rho sw_a q_{irr}.\]  

(2.25)

So:

\[\rho sw_a \Delta W = -D\vec{v}_m + \rho sw_a q_{irr}.\]  

(2.26)

Equation (2.26) again demonstrates that the total work is the sum of displacement work and deformation work. Figure 2.14 shows that the total work by the drag force is always in the sense from material parts to the fluid. A part of the work is dissipated. The displacement work \((-D\vec{v}_m\)), corresponding to mechanical energy increase within the fluid, may either be positive or negative. Displacement work
\((-\vec{D}_{\vec{v}_m})\) is negative for the four configurations in Fig. 2.14. This is the situation that we intuitively expect and that is always correct for stator components. With rotor components, the displacement work of the drag force may be positive. This requires a rather high blade speed compared to the other velocity components (see Exercise 2.5.6). Equation (2.26) demonstrates that the concept of displacement work should be applied with care. Only in the case of a purely active force, without any deformation work associated, as a lift, does the displacement work equal the total work.

### 2.2.8 The Zweifel Tangential Force Coefficient

With an axial cascade, in principle, the component of the force on the blade useful for work is not the lift but the tangential component. It is therefore appropriate to define a coefficient on the basis of \(F_u\) as well. For a constant density fluid, the obvious reference force is \(\frac{1}{2}\rho w^2 c_a\), with \(c_a\) the *axial chord*, as illustrated in Fig. 2.15. When losses are ignored, the pressure at the outlet is lower than the total pressure at the inlet with the amount \(\frac{1}{2}\rho w^2\). The surface between the pressure curves on the pressure and suction sides is the magnitude of \(F_u\). We see the relation with the reference force \((p_{0tr} - p_2)c_a\). This quantity is replaced by the value for lossless constant density flow: \(\frac{1}{2}\rho_2 w^2 c_a\) (\(\rho_2\) is density at outlet for a compressible fluid). This allows the definition of a tangential force coefficient, expressed solely as a function of velocity quantities and the axial solidity \(\sigma_a = c_a / s\), by

![Fig. 2.15 Reference force for the tangential force; left: cascade with moderate flow acceleration (turbine); right: cascade with moderate flow deceleration (pump, fan, compressor)
The concept of tangential force coefficient was introduced by Zweifel in the 1940s. He found that the coefficient was about 0.8 for cascades with good efficiency. The separation values were around 1.1–1.2. Good efficiency means small losses compared to the tangential force. With $C_{Fu} = 0.8$ follows an optimal value for the axial solidity by:

$$\sigma_a = \frac{c_a}{s} = \frac{2}{0.8} \frac{w_a|w_{iu} - w_{2u}|}{w_2^2} = 2.5 \frac{w_a^2 |\tan \beta_1 - \tan \beta_2|}{w_2^2} = \frac{2.5 \cos^2 \beta_2 |\tan \beta_1 - \tan \beta_2|}{w_2^2}. $$

For a compressor with $\beta_1 = -55^\circ$ and $\beta_2 = -35^\circ$, this formula results in $\sigma_a = \frac{c_a}{s} = 1.22$. The chord solidity is $\sigma = \frac{c}{s} = \frac{c_a}{\cos \beta_m} = 1.78$. For a turbine with $\beta_1 = 0^\circ$ and $\beta_2 = -60^\circ$ corresponds $\sigma_a = \frac{c_a}{s} = 1.08$ and $\sigma = 1.43$.

The tangential force coefficient is related to the lift coefficient. Ignoring the effect of losses, the lift coefficient of a cascade is

$$L = \rho |\Gamma| w_m = \frac{1}{2} \rho w_1^2 C_{LI1}.$$

So:

$$C_{LI1} = \frac{2 |\Gamma| w_m}{w_1^2 c} = \frac{2s |w_{2u} - w_{iu}| w_m}{w_1^2 c} = \frac{2 |\Delta w_u| w_m}{\sigma w_1^2}. \quad (2.28)$$

With $c_a / c \approx \cos \beta_m = w_a / w_m$ (for $w_a$ constant), the tangential force coefficient (2.27) is

$$C_{Fu} = \frac{2 |\Delta w_u| w_a s}{w_2^2 c_a} \approx \frac{2 |\Delta w_u| w_m s}{w_2^2 c} = \frac{2 |\Delta w_u| w_m}{\sigma w_1^2} \left(\frac{w_l}{w_2}\right)^2. \quad (2.29)$$

From the comparison between Eq. (2.28) and Eq. (2.29) follows that the tangential force coefficient is a lift coefficient, related to the outlet dynamic pressure. Zweifel’s reasoning thus demonstrates that the lift coefficient with a cascade has to be related to the outlet kinetic energy.

So, for an axial cascade:

$$C_{L2} = \frac{2 |\Delta w_u| w_m}{\sigma w_2^2} \approx C_{Fu} = \frac{2 |\Delta w_u| w_a}{\sigma a w_2^2}. $$
The value of the tangential force coefficient $C_{F_u}$ with present-day, optimally functioning, turbine cascades is about 1.0–1.2. The separation value is 1.4–1.6. These values are the same for constant density and variable density fluids and do not depend on the outlet Mach number for compressible fluids. The tangential force coefficient may very reliably be applied to turbine cascades. With pump, fan and compressor cascades, application of $C_{F_u}$ is possible as well. The optimal value is about 1.0–1.2 as well, but values show a much larger spreading than with turbines (see next section).

### 2.2.9 The Lieblein Diffusion Factor

With strongly decelerating cascades, the lift potential is also determined by the inlet velocity level. Therefore, the lift coefficient $C_{L_{2*}}$, but as well $C_{L_{1*}}$, is not universally applicable. Figure 2.16 shows typical velocity distributions near the blade surface (boundary layer edge) of a strongly accelerating turbine cascade and a strongly decelerating compressor cascade. The figure demonstrates that the tangential force with the turbine is strongly correlated to the outlet dynamic pressure. The correlation is much weaker with the compressor cascade, due to the strong acceleration downstream of the leading edge stagnation point, both on suction and pressure sides. Consequently, the tangential force is also determined by the value of the inlet dynamic pressure. Deceleration is limited with fans and pumps and the tangential force coefficient can be applied more reliably than with compressors. When studying axial turbines, axial fans and axial pumps, the tangential force coefficient, according to Eq. (2.29) may be applied. With compressor cascades, the concept of the diffusion factor is mostly used to determine the lift capacity of a cascade. This concept was introduced by Lieblein in the 1950s and there have been some variants. A diffusion factor is the ratio of the deceleration at the suction side $(w_{max} - w_2)$ to a reference velocity, mostly $w_1$. The diffusion factor in that sense will be discussed in Chap. 13 (axial compressors). With axial and radial fans and pumps, the concept of local diffusion factor may be used as well. It is the ratio of $(w_{max} - w_2)$ to $w_{max}$.

![Fig. 2.16 Velocity distribution near the blade surface in a highly loaded turbine cascade (left) and a highly loaded compressor cascade (right)](image-url)
A criterion for avoidance of separation is that this factor should be limited to 0.5. This means that \((w_{\text{max}} - w_2)\) must not exceed \(w_2\). This criterion may also be applied to estimate the lift capacity of a cascade. We will use the local diffusion factor for radial cascades in Chap. 3. So:

\[
DF = \frac{w_{\text{max}} - w_2}{w_1} \quad \text{and} \quad D_{\text{loc}} = \frac{w_{\text{max}} - w_2}{w_{\text{max}}}.
\]

### 2.2.10 Performance Parameters of Axial Cascades

The losses within the cascade are expressed by the total pressure drop \((p_{\text{in}} - p_{\text{out}})\), relative to the inlet dynamic pressure \(q_1\) with a decelerating cascade or to the outlet dynamic pressure \(q_2\) with an accelerating cascade. Literature offers correlations to determine loss coefficients. Methods differ for decelerating and accelerating cascades. The deviation angle \(\delta\) can be determined, as well as the optimal value of the angle of incidence \(i\). The study of these correlations exceeds the objective of the present book. We further use simple correlations. We refer to Dixon and Hall [2], Japikse and Baines [4] for more complete correlations concerning cascades.

### 2.3 Channels

#### 2.3.1 Straight Channels

For a straight channel with constant cross-section area \(A\) and with fully developed flow (velocity profile does not change in flow direction), the relation between the pressure drop (denoted here by \(-\Delta p\)) and the shear stress on the wall is

\[
A(-\Delta p) = \tau_w O L,
\]

with \(O\) being the circumference and \(L\) the length of the part considered. Thus

\[
\frac{-\Delta p}{\rho} = \frac{\tau_w}{\rho} \frac{O}{A} L = \frac{\tau_w}{\rho} \frac{4L}{D} \left(\frac{1}{2} \bar{v}^2\right).
\]

\(D\) is the hydraulic diameter \((D = 4A/O)\) and \(\bar{v}\) the average velocity. The term \(-\Delta p/\rho\) represents the mechanical energy loss \(-\Delta E_m\). A loss coefficient is defined by

\[
\xi = \frac{-\Delta E_m}{\frac{1}{2} \bar{v}^2} = f \frac{L}{D},
\]

(2.30)
where $f$ is the Darcy friction factor. The Fanning friction factor is applied sometimes, defined by $\tau_w / (\frac{1}{2} \rho v^2)$. There is a ratio 4 between both factors. Figure 2.17 shows schematically the well-known Moody diagram for ducts with a circular section. The Darcy friction factor is represented. The full diagram is published in almost all books on basic fluid mechanics. It can also be found on many internet sites (look for Moody friction factor chart).

The Colebrook-White equation renders the friction factor for turbulent flows:

$$\frac{1}{\sqrt{f}} = -2\log \left[ \frac{2.51}{Re} \sqrt{f} + \frac{k}{3.7D} \right].$$

The Reynolds number is defined by $Re = \nu D / \nu$, with $\nu$ being the kinematic viscosity coefficient. Roughness is represented by the equivalent sand-grain roughness $k$.

The implicit equation may be replaced with a very good approximation (better than 1.5%) by the explicit equation of Haaland [3]:

$$\frac{1}{\sqrt{f}} = -1.8\log \left[ \frac{6.9}{Re} + \left( \frac{k}{3.7D} \right)^{1.11} \right].$$

Channels in turbomachines never have a constant cross section and a fully developed flow. It is customary however to estimate friction losses with the Moody-diagram on the basis of an average length and an average hydraulic diameter. Therefore, the applied method only gives an approximation.
2.3.2 Bends

Flow in a bend changes pressure and velocity distributions, generates adverse pressure gradients and secondary flows and the curvature affects the turbulence.

Figure 2.18 sketches the velocity distribution with an ideal fluid (frictionless). The centrifugal force due to flow curvature generates a static pressure increase at the bend outer part. Velocity is lower at the outer than at the inner part. There is an adverse pressure gradient at the entrance of the outer part and at the exit of the inner part of the bend. The effect of friction causes the velocity to be higher within the flow core than at the walls. The consequence is that the centrifugal force due to bend curvature is higher within the flow core. The difference generates two vortex flows as sketched in Fig. 2.19. This transverse flow is termed secondary flow. Low-energy fluid migrates from the outer part of the bend to the inner part. With 45° bends, with ratios of the radius of curvature of the bend to the diameter of 1–3, which are common values, it comes out that low-energy fluid arrives in the adverse pressure gradient zone at the bend exit. There is then a separation risk. With larger bend angles, core fluid arrives in the adverse pressure gradient zone at the bend exit. There is thus a far lower separation risk with a 90° degree bend, which is a surprising observation. Bends further affect the turbulence. Higher and lower velocity turbulent eddies occur simultaneously at a certain place. Eddies with high instantaneous velocity are subjected to high centrifugal force. These eddies migrate to the bend outer part. They create vortex motions in the boundary layer that break down into turbulence. The consequence is that turbulence at the outer part of the bend increases. Inversely, eddies with low instantaneous velocity migrate to the inner part and damp the turbulence there. This phenomenon of turbulence migration is clearly observed [6], but is not fully understood yet. We will describe it with the term turbulence segregation. A consequence of the segregation is weakening of the boundary layer at the inner part. This further increases the boundary layer separation risk at the bend inner part in the exit region.

Bend phenomena occur intensively in centrifugal rotor channels. At the inlet, flow is deflected from the axial into the radial direction. Deflection also occurs within the rotor channels, and the Coriolis force generates a similar segregation effect as the centrifugal force. A bend represents a supplementary loss, on the one hand because of increased friction within the bend, on the other by flow homogenisation downstream of the bend. By homogenisation we mean here the recovery process of the velocity profile towards an equilibrium profile adapted to the downstream channel. Velocity rearrangement of the fluid layers produces vortices that break down into turbulent eddies. These eddies interact and further break down to smaller size. The smaller the turbulent motion, the greater the impact of viscosity forces onto the motion and the greater the fraction of the energy dissipated in heat during the breakdown from larger to smaller structures. This process, causing energy dissipation, is termed the energy cascade. Figure 2.20 illustrates the dissipation process associated to velocity profile recovery.
after a sudden expansion. The recovery after a bend is similar. With a bend of 90° and $r/d = 2$, the bend loss coefficient amounts to 0.16. About half of the loss occurs in the bend itself and about half is due to homogenisation after the bend.

At the inlet of a centrifugal machine, the losses due to homogenisation are not assigned to the inlet, as the inlet is not followed by a duct in which homogenisation could occur. So, the order of magnitude of the loss coefficient due to the bend is about 0.1, but the inhomogeneous inlet flow of the rotor reduces its efficiency. In the same way, homogenisation loss downstream of the rotor is not assigned to the rotor. A mixing space may occur after the rotor. Mixing losses are calculated for this space. If there is a diffuser immediately downstream of the rotor, its efficiency is impaired due to the inhomogeneous flow. The effect of inhomogeneous inflow with a diffuser will be discussed in the next section.
2.4 Diffusers

2.4.1 Dump Diffusers

Figure 2.20 sketches the flow in a sudden expansion of a channel. Flow is decelerated. A sudden expansion thus functions as a diffuser. The figure also illustrates the loss mechanism by mixing. The loss is the kinetic energy related to the velocity difference (see Exercise 2.5.4):

\[
q_{\text{irr}} = \frac{(v_1 - v_2)^2}{2} = \left(1 - \frac{v_2}{v_1}\right)^2 \frac{v_2^2}{2} = \left(1 - \frac{A_2}{A_1}\right)^2 \frac{v_2^2}{2} = \xi v_2^2.
\]  

(2.31)

Diffusion by sudden expansion or dump diffusion is efficient with a rather small area ratio. For instance, \(A_2/A_1 = 1.25\) gives \(\xi = 0.04\). A higher area ratio is useful as well. For instance, \(A_2/A_1 = 2\) gives \(\xi = 0.25\). A loss coefficient with value 0.25 is not disadvantageous for a diffuser, as will be discussed below. That is why dump diffusion is applied in turbomachines, which is rather surprising at a first glance. The main advantage of dump diffusion is its realisation in a short distance.

2.4.2 Inlet Flow Distortion

Diffusers are channels where flow is decelerated and dynamic pressure is converted into static pressure. This process is called pressure recovery. The attainable pressure recovery or diffusion strongly depends on the uniformity of the incoming flow. Figure 2.21 compares the diffusion with uniform and non-uniform incoming flow of an ideal fluid. In both cases, 50% dynamic pressure recovery based on the average inlet velocity is intended. The figure shows the position where the 50% recovery is obtained. With the uniform flow, the corresponding velocity is \(\sqrt{1 - 0.5} \approx 0.71\) times the inflow velocity and the corresponding section area is 1.41 times the inflow section area. With a non-uniform inlet, a larger area ratio is required, so a larger covered length with a given opening angle, due to the much faster velocity decrease of the slower part of the flow. At inflow, the velocities are 1.25 and 0.75 times the average velocity. The necessary reduced velocities are \(\sqrt{(1.25)^2 - 0.5} \approx 1.03\) and

---

Fig. 2.21 Diffusion with uniform and non-uniform inlet flow
\[
\sqrt{(0.75)^2 - 0.5} = 0.25
\]
times the average inflow velocity. The corresponding section area is 2.11 times the inflow section area. The conclusion is that disturbance of the inlet flow uniformity causes a reduction of the possible pressure recovery. Length limitation normally occurs with turbomachines, generating an inherent limitation of the pressure recovery.

Within a real flow, the velocity at the walls equals zero. The slow flow at the walls can only participate in the diffusion process when there is momentum transfer from the core flow. This transfer is due to molecular viscosity and to turbulent mixing. Both processes are coupled with energy dissipation. In the first phase of a diffusion process, strong deceleration near the walls occurs. This results in a shear stress zone moving to the centre as the fluid advances through the diffuser. Turbulence produced by the shear enhances momentum exchange. At the outlet of diffusers with a limited length, the core flow is nearly unaffected. Figure 2.22 illustrates the inhomogeneous velocity pattern at the outlet caused thereby. Velocity has only decreased little within the flow core. From that we infer that insufficient possibility for deceleration within the core flow, more than energy dissipation, constitutes the main limitation of the pressure recovery. Figure 2.22 also shows that the loss mainly consists of mixing loss downstream of the dif-

---

**Fig. 2.22** Velocity profile evolution within a diffuser with limited length

---

**Fig. 2.23** Diffusion with separation; *left*: large separation zone; *right*: reduced separation zone by combination with sudden expansion

---

**Fig. 2.24** Postponed diffusion after a bend and diffusion concentration on the high-energy side
The flow structure is not highly different from that with dump diffusion. Diffusion within a gradually widening channel so intrinsically is a process with high losses.

### 2.4.3 Flow Separation

Separation occurs when the divergence of a diffuser is too strong. Pressure recovery within a diffuser with large separation is low and the exit flow is heavily deformed, with backflow at some places. Large-scale backflow is illustrated in Fig. 2.23 (left). It occurs with strong divergence and is nearly steady. The flow pattern is denominated *steady stall*. With less strong divergence, the separation is typically time dependent. Separation zones then build up repeatedly inside the diffuser and are moved out by convection. This growth and removal pattern is termed *transitory stall*. When diffusers function in transitory stall, large fluctuations in pressure recovery and outlet velocity distribution occur. A separation zone with strong backflow, as in Fig. 2.23 (left), causes large mixing losses downstream of the diffuser. It might then be more advantageous to combine a diffuser with a lower opening angle with a dump diffuser, as shown in Fig. 2.23 (right).

### 2.4.4 Flow Improvement

As the efficiency of a diffuser depends strongly on the uniformity of the velocity at the inlet, efficient diffusion might be impossible if the inlet velocity varies too much. Figure 2.24 shows, as an example, a diffuser downstream of a bend. It is then better to postpone the diffusion in order to reduce the distortion of the profile and to concentrate the divergence of the diffuser on the higher energy side.

Diffuser efficiency may be further improved as illustrated in Fig. 2.25. Very efficient is removal of low-energy flow at the wall by suction (Fig. 2.25 left, top wall). A somewhat similar approach is energy addition by injection in the boundary layers by tangential slots (Fig. 2.25 left, bottom wall). These techniques are applied only under special circumstances as with wind tunnel diffusers or when the fluid is drained for secondary purposes. A simple method consists in mounting guiding surfaces in a diffuser with a wide opening angle. This is mostly the only practical method to improve the outlet flow distribution (for dimensioning see 2.4.6).

![Fig. 2.25](image-url) Removal of low-energy fluid (*left; top wall*); injection of high-energy fluid (*left; bottom wall*); application of guiding surfaces (*right*)
Mounting guiding surfaces is easy with two-dimensional diffusers, but somewhat more complex with conical diffusers.

### 2.4.5 Representation of Diffuser Performance

Geometrical characteristics are shown in Fig. 2.26. These are the inlet and outlet areas $A_1$ and $A_2$, the length $L$ and the hydraulic radius $R$ of the inlet section ($R_1$ for a circle; $H_1$ for rectangle with high aspect ratio). A common way to represent pressure recovery is shown in Fig. 2.27[6]. Lines of constant pressure recovery are plotted in a diagram with diffuser length to inlet radius ratio in the abscissa and the area ratio in the ordinate.

Pressure recovery is expressed with a coefficient $C_p$ defined by

$$C_p = \frac{p_2 - p_1}{\frac{1}{2} \rho \bar{V}_1^2}.$$ 

The loss coefficient, in principle, is

$$\xi = \frac{p_{01} - p_{02}}{\frac{1}{2} \rho \bar{V}_1^2} = I - \left(\frac{\bar{V}_2}{\bar{V}_1}\right)^2 - C_p = C_{p, id} - C_p,$$

where $C_{p, id}$ is the ideal pressure recovery coefficient:

$$C_{p, id} = I - \left(\frac{A_1}{A_2}\right)^2.$$ 

For a diffuser with a free outlet, outlet kinetic energy is a loss as well, and so

$$\xi = \frac{p_{01} - p_2}{\frac{1}{2} \rho \bar{V}_1^2} = \frac{p_1 + \frac{1}{2} \rho \bar{V}_1^2 - p_2}{\frac{1}{2} \rho \bar{V}_1^2} = I - C_p.$$ 

![Fig. 2.26 Geometrical characteristics of diffusers](image)
The loss coefficient of a diffuser with a free outlet may be described again as 
\[ C_{p,\text{id}} - C_p \], with the ideal pressure recovery coefficient being unity.

The meaning of the dashed lines 1 and 2 in the diagram of Fig. 2.27 is that they separate the different flow regimes. In the area on the right of line 2, where the pressure recovery lines are nearly horizontal, flow is stable. With a constant area ratio, pressure recovery decreases with larger length due to increase of the friction surface. Diffusers in this area are unnecessarily long. Thus, for given area ratio, the most efficient diffusers with stable flow are just on line 2. Above line 1 steady stall occurs. For a given length, pressure recovery decreases with increasing area ratio. So, it is useless to operate a diffuser in this area. In the area between the two lines, transitory stall occurs. Most diffusers applied in practice have geometries in this region. Line 1 in Fig. 2.27 is approximately the surface ratio corresponding to minimum total pressure loss at a given length ratio (vertical tangents to the contour lines of \( C_p \)). Line 2 is approximately the length ratio corresponding to minimum total pressure loss at a given surface ratio (horizontal tangents to the contour lines of \( C_p \)). So, mostly, the lines separating the flow regimes are not drawn in a diffuser diagram and the user is supposed to know. In turbomachinery applications, where length limitation always occurs, in principle, the most efficient operation is for surface ratio corresponding to minimum total pressure loss at a given length ratio, thus on line 1. But this operation implies heavy unsteady stall or even steady stall. So, usually, for limiting oscillations, it is better to choose the geometry of a diffuser somewhere in the middle of the zone where the contour lines of pressure recovery change from horizontal to vertical direction.

Fig. 2.27 Performance of conical diffusers with a uniform inlet profile and a free outlet; adapted from Miller [6]
Figure 2.27 shows the pressure recovery with a uniform incoming flow. The corresponding diagram for a fully developed inflow is shown in Fig. 2.28 [6]. Both diagrams are valid with Re = 10^6. A minor correction for the change of the Reynolds number is required. Figure 2.29 shows the performance of conical diffusers connected to circular ducts at inlet and outlet, with fully developed inlet flow. For large length, the contour lines are similar to those of the previous diagrams. Differences occur with short dimensionless lengths.

The mixing after diffusion in a constant section pipe results in a static pressure increase. The length of the duct required to obtain complete mixing depends on the area ratio and the diffuser angle but typically is about 4 diameters. After the maximum pressure position, the friction factor remains somewhat higher than that of a fully developed flow during 20–50 diameters. The loss coefficient for this additional loss is about 0.1. The pressure recovery shown in Fig. 2.29 is the pressure recovery at the maximum pressure position.

### 2.4.6 Equivalent Opening Angle

A diffuser may be characterised by a half opening angle. For a conical diffuser, this is

\[ tg \alpha = \frac{R_2 - R_1}{L}. \]
For a general diffuser the same formula is used with the hydraulic radius and the term equivalent opening angle is employed. From the performance diagrams 2.27–2.28 follows that optimum operation approximately corresponds to an opening angle $2\alpha = 18^\circ$ to $8^\circ$, for the length ratio varying from small to large. If the diffuser length that is available for the desired velocity reduction, i.e. the area ratio, is too small, either a diffuser followed by a sudden expansion (Fig. 2.23) or guiding surfaces (Fig. 2.25) should be applied. The number of guiding surfaces should be determined so that the equivalent opening angle criterion is met for each partial diffuser.

2.4.7 Diffusion in a Bend

All phenomena occurring in bends and straight diffusers occur in a still more extreme form in a bent diffuser. By adding a bend immediately downstream of a diffuser, the adverse pressure gradient at the outward part of the bend entrance increases (see Fig. 2.18) and there is high risk for separation. Addition of a diffuser downstream of a bend steepens the pressure gradient at the inward part of the exit of the bend (see Fig. 2.18) and makes separation very likely. From the above
examples it is clear that diffusion within a bend is delicate. The phenomena discussed before, such as secondary flow and segregation of turbulent eddies, strongly intervene. However, a design in which bend phenomena and diffusion are well matched allows a pressure recovery that is far superior to that of a serially connected bend and a straight diffuser. The crucial aspect is useful application of secondary flow in order to bring high-energy fluid to the places with imminent separation. The optimum geometry strongly depends on the inlet conditions. The consequence is that there is no universal optimum geometry. For instance, bent diffusers are optimised for vertical shaft hydraulic turbines. Figure 2.30 shows an example (Francis turbine). A pressure recovery up to 75% is attained. The optimisation requires computational techniques and turbulence models. At present, optimisation of stationary diffusers is very well possible. It is more difficult however for rotating diffusing channels (radial pumps and compressors), as complex secondary flow patterns intervene. Notice that a large pressure recovery requires a sufficient covered length (Fig. 2.30). Most radial pumps, fans and compressors have limited rotor channel lengths. Hence, the earlier mentioned limit of velocity deceleration ratio $w_2/w_1$ of about 0.8–0.9 in order to avoid separation. In some radial compressors, rotor channels with greater lengths are applied, by mounting an inducer (see Chap. 14). A velocity ratio $w_2/w_1$ as low as 0.6 may then be realised, but the pressure recovery is not very good, however. Figure 2.24 is intended as a practical suggestion to combine a bend with a diffuser without applying means of optimisation. It is advisable to mount the bend first, and then a duct segment with constant section (length about one diameter) in which mixing occurs by secondary flow, followed by an asymmetrically mounted diffuser.
2.5 Exercises

2.5.1. The figure sketches laminar flow between a moving block and a stationary flat wall. The block moves parallel to the wall at velocity $v$. There is no pressure difference in the flow direction in the space between the block and the wall. Reason that the shear stress $\tau$ within the shear layer is constant. Demonstrate that dissipated work per surface unit and per time unit equals displacement work $(v \cdot \tau)$ exerted by the object onto the shear flow. Demonstrate that this result remains valid with turbulent flow.

![Diagram of laminar flow between a moving block and a stationary wall.]

2.5.2. The left figure sketches the velocity profiles in a 2D-channel with fully developed laminar flow of an incompressible fluid ($\rho=\text{constant}, \ m=\text{constant}$). Fully developed flow means that no change of velocity profile occurs in the flow sense. Derive that the shear stress varies linearly from the wall value $\tau_w$ to zero in the channel centre. Demonstrate that the velocity profile is a parabola. Derive that the energy dissipated per wall surface unit and per time unit equals $U_m \tau_w$, with $U_m$ being the average velocity within the channel. Note that for attaining these results nothing more is necessary than that the shear stress varies linearly along the height. So argue that the same result is attained with a turbulent profile (central figure) ($\mu \neq \text{constant}$). The right figure sketches the velocity and shear stress profiles with a turbulent flow of a boundary layer over a flat plate at a zero pressure gradient. The boundary layer grows. So the flow is not fully developed. The shear stress profile differs little from a linear one. So, the former result is still valid with a good approximation.

![Velocity profiles for laminar and turbulent flow.]

2.5.3. The figure sketches the mixing of two flows with velocities $3/2 \ v$ and $1/2 \ v$ within a 2D-channel. Both flows occupy half the section. Velocity therefore is $v$ after complete mixing. Determine the pressure increase by complete mixing. Determine the energy dissipation by mixing per mass flow rate unit, ignoring friction.
2.5.4. The figure sketches a flow with sudden expansion (dump diffusion). Determine the pressure increase and the energy dissipation per mass flow rate unit.

\[ \Delta p = \frac{\Delta p}{\rho} = \frac{1}{2} \rho \left( \frac{v_2^2}{2} - \frac{v_1^2}{2} \right) \]

\[ q_{\text{irr}} = \frac{1}{4} \frac{v_1^2}{2} \]

2.5.5. The figure is a sketch of a jet pump. Water is injected from an external source at velocity \( v_1 \) into a tube, driving water from the suction pipe to velocity \( v_2 \) at the injector outlet position. Determine the pressure increase in the driven flow. Determine the pump characteristic as the mechanical energy increase of the driven flow \( \Delta E_{m2} = (p_{03} - p_{02}) / \rho \) as a function of its mass flow rate at given values of \( A_1, A \) and \( v_1 \). Define surface ratio \( \alpha = A_1 / A \) and velocity ratio \( U = v_1 / v_2 \). Take \( \alpha = 0.4 \) as an example and consider \( U \) as a mass flow rate measure. Define a pressure coefficient \( \psi = \Delta E_{m2} / (v_1^2 / 2) \).
Study the energy exchange. An expression for the energy increase in the driven flow has already been formulated. Determine the mechanical energy decrease in the driving jet flow \((p_{01} - p_{03}) / \rho\). In most applications, the driving water for the jet is tapped from the flow in the pressure pipe. Driving water is then produced with a pump increasing the total pressure \(p_{03}\) to the total pressure \(p_{01}\). With an ideal pump, the energy increase required exactly equals the mechanical energy decrease within the driving jet between the injection point (1) and the jet pump outlet (3). The ratio of the energy increase in the driven flow to the energy decrease in the driving flow may then be defined as the efficiency \(\eta\) of the jet pump. Note that this efficiency definition also applies with driving water from another origin.

Determine the efficiency \(\eta\) for varying \(\alpha\) and \(U\). Study the values of \(\alpha=0.2, 0.4, 0.6, 0.8\) combined with the values of \(U=0, 0.25, 0.50, 0.75, 1\). Observe that the efficiency is weakly dependent on \(\alpha\), but strongly dependent on \(U\). To attain a good efficiency (better than 0.75), \(U\) must exceed about 0.6. This implies that the velocity difference between driving and driven flow at the mixing position must not be too big. This is obvious, as the loss considered is mixing loss. The driven jet must thus be highly accelerated at the position of the injector. See the geometry of a real jet pump as shown in the figure below. The velocity of the driven jet must mostly be limited in order to avoid cavitation. For instance, to \(v_2 = 10 \text{ m/s}\) corresponds a pressure decrease in the suction pipe of \(\rho v_2^2 / 2 = 50000 \text{ Pa} = 0.5 \text{ bar}\). Such a strong pressure decrease implies that the geometrical suction height must be under 5 m. Jet pumps are therefore mostly mounted below the water surface in the suction well (submerged). A pressure drop of 0.5 bar is then no problem. A diffuser beyond the mixing chamber is required as well. In the example quoted, velocity exceeds 10 m/s after mixing, but the velocity in a pipe is typically at maximum about 2 m/s. So the diffuser generates substantial losses. Attaining a considerable pressure increase with good efficiency is thus impossible.

For the study of pressure increase and mass flow rate in the driven flow, we define still other parameters. As the measure for the pressure increase, we define the ratio \(\delta\) of the total pressure increase in the driven flow to the total pressure decrease in the driving flow. As the measure for the mass flow rate, we define the ratio \(\mu\) of the mass flow rate of the driven flow to the mass flow rate of the driving flow. With these definitions, the efficiency is \(\eta = \mu \delta\). Determine \(\delta\) and \(\mu\) for varying \(\alpha\) and \(U\). Study again the values of \(\alpha=0.2, 0.4, 0.6, 0.8\) combined with the values of \(U=0, 0.25, 0.50, 0.75, 1\). Observe that, for a similar efficiency, as well a low mass flow rate together with a high pressure increase as a high mass flow rate together with a low pressure increase may be chosen. The choice of the combination is mainly determined by the parameter \(\alpha\) with only a weak effect of the value of \(U\). Since the analysis takes neither the friction loss in the mixing chamber nor the diffuser loss into account, the velocity ratio should be chosen lower in practice than the values obtained from the analysis. Then, velocities in the mixing chamber and in the diffuser decrease. A practical value of \(U\) is about 0.5. The efficiency obtained in the analysis is about 0.65, arriving at about 0.35 in practice.
2.5.6. The figure sketches a moving linear cascade of cylindrical rods. The oncoming flow has velocity \( v_1 \). The speed of the rods is \( u \). The ratio of the blade speed to the oncoming flow velocity is termed the *speed ratio*, being here \( \lambda = u / v_1 = 8 \). The solidity of the cascade is \( \sigma = d/s \), where \( d \) is the diameter of the rods. We choose here \( \sigma = 1/8 \), assuming a drag coefficient \( C_D = 1 \). Determine the velocity downstream of the rods. Determine the work done on the fluid. Determine mechanical energy increase within the flow and the energy dissipated. With cascade analysis it is customary to express the velocity components at the position of the cascade in proportion to the oncoming velocity and the blade speed. The axial and tangential velocity components of a driven cascade are represented by 

\[
\begin{align*}
\omega_a &= v_1(1 + a), \\
\omega_{mu} &= -u(1 - b)
\end{align*}
\]

The factors \( a \) and \( b \) are called *interference factors*. With a linear cascade, as sketched in the figure, \( a = 0 \).

\[
A : b = 0.173, v_{2u} / u = 2b, \Delta W / u^2 = 2b = 0.346, \Delta E_m / u^2 = 0.053, q_{irr} / u^2 = 0.293
\]

2.5.7. The figure is a sketch of an annular streamtube with an infinitesimal height through the rotor of an axial fan. The rotor blade speed within the section of the streamtube is \( u \). As speed ratio we choose \( \lambda = u / v_a = 3 \), being a typical value for a half radius section (tip value \( \lambda_T = 6 \)). Applying an interference factor, as in the previous exercise, we set \( \omega_{2u} = -u(1 - 2b) \). The solidity of the cascade is \( \sigma = c/s = 2/3 \). We choose \( C_L = 1 \) as the lift coefficient, ignoring the drag, so \( C_D = 0 \). Determine the work done on the fluid. Determine the static pressure increase across the rotor. What is the degree of reaction? What is the static pressure increase obtained by the fan by addition of a stator turning the velocity into the axial direction?

\[
A : b = 0.152, \frac{\Delta W}{u^2} = 2b = 0.304, \frac{\Delta (p/\rho)}{u^2} = 2b - 2b^2 = 0.258,
\]
The figure is a sketch of an annular streamtube with an infinitesimal height through a wind turbine rotor. The velocity far upstream is $v_0$. The rotor blade speed within the streamtube section is $u$. As speed ratio we choose $\lambda = u/v_0 = 3$, being a typical value for a half radius section (a typical tip value is $\lambda_T = 6$; see Chap. 10 on wind turbines).

Due to the power extraction, the velocity within the streamtube decreases. With interference factors, we set $v_{1a} = v_{2a} = v_0(1 - a)$ en $w_{2a} = -u(1 + 2b)$. Positions 1 and 2 are situated immediately upstream and downstream of the blade segment. The components of the average velocity on the blade segment are $w_{ma} = v_0(1 - a)$ and $w_{mu} = -u(1 + b)$. Cascade solidity $\sigma = c/s = 1/12$. We take $C_L = 1$ as the lift coefficient and ignore the drag, thus $C_D = 0$. Determine the velocity immediately upstream and downstream of the rotor within the streamtube considered. Determine the power transferred. Assume, as with the one-dimensional propeller analysis in Chap. 1 (Exercise 1.9.5), that there is symmetry in the pressure variation within the streamtube upstream and downstream of the rotor, so that the resulting pressure force onto the streamtube envelope into the axial direction equals zero.
\textbf{A:} The momentum balance (no pressure force in axial direction) into the axial direction on the streamtube results in

\[ -L_a = \rho s v_{1a} (v_{3a} - v_0). \]  

(2.32)

Momentum into the axial direction locally results in

\[ L_a = (p_1 - p_2)s. \]

Combination gives

\[ \frac{p_1 - p_2}{\rho} = v_{1a} (v_0 - v_{3a}). \]  

(2.33)

The work equations within the streamtube upstream and downstream of the rotor are

\[ \frac{p_a + \frac{v_0^2}{2}}{\rho} = \frac{p_L + \frac{v_L^2}{2}}{\rho}, \quad \frac{p_2 + \frac{v_2^2}{2}}{\rho} = \frac{p_a + \frac{v_2^2}{2}}{\rho}. \]

From that:

\[ \frac{p_1 - p_2}{\rho} = \frac{v_2^2 - v_L^2}{2} + \frac{v_0^2 - v_3^2}{2} = \frac{v_{2u}^2 - v_{3u}^2}{2} + \frac{v_{2u}^2 - v_{3u}^2}{2}. \]  

(2.34)

Combination of (Eq. 2.33) and (Eq. 2.34) results in

\[ v_{1a} (v_0 - v_{3a}) = \frac{v_0 + v_{3a}}{2} (v_0 - v_{3a}) + \frac{v_{2u}^2 - v_{3u}^2}{2}. \]

Due to the radius increase within the streamtube downstream the rotor, \( v_u \) decreases. Thus, \( v_{3u} \) is smaller than \( v_{2u} \). We ignore the difference between \( v_{3u} \) and \( v_{2u} \) here. We accept that in (Eq. 2.34) \( p_1 - p_2 \) becomes somewhat smaller. This is equivalent to the assumption that some losses occur within the flow downstream of the rotor. With this simplification follows

\[ v_{1a} = v_{2a} = \frac{1}{2} (v_0 + v_{3a}). \]

The flow deceleration is then, as with the one-dimensional analysis, the same upstream and downstream of the rotor.

We set \( v_{1a} = v_{2a} = v_0 (1 - a) \), \( v_{3a} = v_0 (1 - 2a) \), \( v_{2u} = -u (2b) \).

The energy extraction in the streamtube according to (Eq. 2.34), ignoring the difference between \( v_{2u} \) and \( v_{3u} \), is

\[ \frac{p_1 - p_2}{\rho} + \frac{v_1^2 - v_2^2}{2} = \frac{v_0^2 - v_{3a}^2}{2} - \frac{v_{2u}^2}{2} = \frac{v_0^2}{2} (4a - 4a^2) - \frac{u^2}{2} (4b^2). \]  

(2.35)
The energy extraction according to Euler’s formula is
\[ \Delta W = u(v_{1u} - v_{2u}) = u(2bu). \]  
(2.36)

Equality of (Eq. 2.35) and (Eq. 2.36) results in
\[ b\lambda^2 = a(1-a) - b^2\lambda^2. \]  
(2.37)

The momentum balance in the tangential direction locally on the blade segment is
\[ -L_u = \rho v_{1a}s(w_{2u} - w_{1u}) = \rho v_{1a}sv_{2u} = -\rho v_{1a}s2bu. \]  
(2.38)

The velocity triangle at the position of the blade segment is sketched in the figure:

From this:
\[ L_u = L\cos\beta_m = C_L\frac{1}{2}\rho w_m^2c\frac{v_{1a}}{w_m} \]  
(2.39)

Combination of Eq. (2.38) and Eq. (2.39) results in
\[ v_{1a}s2bu = \frac{1}{2}C_Lcv_{1a}w_m, \]

or
\[ 2b = \frac{\sigma C_L}{2}\left(\frac{(1-a)^2}{\lambda^2} + (1+b)^2\right). \]  
(2.40)

With \( A=3 \) and \( sC_L=1/12 \), (Eq. 2.37) and (Eq. 2.40) result in: \( a=0.2700, b=0.0214. \)

The power exchanged per surface unit (Eq. 2.34) is
\[ \rho v_0(1-a)2bu^2 = \rho \frac{v_0^3}{2}(1-a)4b\lambda^2 = \rho \frac{v_0^3}{2} \times 0.5624. \]

2.5.9. Verify the tangential force coefficient (Zweifel) of the steam turbine cascades shown in Fig. 1.11 of Chap. 1.

A: \( R=0 \): stator \( C_{fu}=0.50 \) (low load), rotor \( C_{fu}=0.90 \) (moderate load); \( R=0.50 \), stator and rotor: \( C_{fu}=0.50 \) (low load).

2.5.10. A cylindrical tube should undergo a diameter increase with a factor 2. For this increase, there is only a length of 1 diameter available. Design an optimum
Basic Components

A diffuser for this application with the Fig. 2.29 diagram. What is the realisable pressure recovery?

A: The most advantageous solution is a conical diffuser with an optimum opening angle (area ratio equal to 2), followed by a sudden expansion. The loss coefficient of the conical diffuser, $C_{p/id} - C_p$, is about 0.135 according to Fig. 2.29. The loss coefficient of the sudden expansion is 0.25 according to Eq. 2.31 (in reality somewhat higher due to inflow irregularity). The resulting loss coefficient based on inlet kinetic energy is $0.135 + 0.25/4 = 0.2025$. $C_{p/id} = 0.9375$. $C_p = 0.735$. This solution is more elegant than with guiding surfaces according to Fig. 2.25, which would require a very small opening angle of the partial diffusers (L/R about 16 according to Fig. 2.29). The pressure recovery of a conical diffuser with area ratio 4 is as low as for a sudden expansion: $C_p = 0.375$.

2.5.11. A diffuser with a free outlet has $L/R_1 = 2$ and $AR = 1.5$. Determine, with Figs. 2.27 and 2.28, the increase of the pressure recovery coefficient when the inlet profile is transformed from fully developed into uniform.

A: $C_p$ goes from 0.45 to 0.50.

2.5.12. The figure shows a Poncelet-type waterwheel. This is an undershot waterwheel with curved blades. The water accelerates under the slide due to the height difference ($v_1^2 / 2 = gh$, ignoring losses). The direction of the water jet generated under the slide forms the angle $\alpha$ with the tangential direction at the periphery of the wheel. The wheel speed at the periphery ($u$) is allowed to vary. Magnitude and direction of the relative velocity at the wheel inlet change by this. We assume that the blade angle at inlet $\beta$ is adapted so that the jet enters the wheel perfectly aligned with the blade direction (with a given machine, this condition is only correct for one operating point; thus we consider a design optimisation with a blade angle not given a priori). We assume that the water leaves the wheel at the same angle as at the inlet, so that $w_2 = w_1$.

Determine a formula for the rotor work at fixed values for $v_1$ and $\alpha$ and a variable value for $u$. Express the internal efficiency of the wheel, ignoring all friction losses. Determine at what peripheral speed the maximum efficiency occurs. Determine the velocity triangle at the outlet together with a formula for the outlet velocity $v_2$. Interpret the efficiency attained, in other words determine which loss occurs. Conclude from the result that a small $\alpha$ is useful (in practice $\alpha = 15^\circ$ is applied). Is there still another apparent loss mechanism?
A: The figure sketches the velocity triangles at the inlet and the outlet of the rotor.

\[
\begin{align*}
\Delta W &= u(w_{1u} - w_{2u}) = 2u(v_1 \cos \alpha - u) \\
&= 2v_1 \cos \alpha - 2u
\end{align*}
\]

Maximum work is attained by \( u = \frac{v_1 \cos \alpha}{2} \).

Maximum work is \( (\Delta W)_{\text{max}} = \frac{v_1^2}{2} \cos^2 \alpha \) with \( \frac{v_1^2}{2} = gh \).

So:

\[
\eta_i = \frac{\Delta W}{gh} \quad \text{and} \quad (\eta_i)_{\text{max}} = \cos^2 \alpha
\]

In the inlet triangle, then \( w_{1u} = \frac{v_1 \cos \alpha}{2} = u \).

In the outlet triangle, then \( w_{2u} = -u \).

The outlet triangle is orthogonal: \( v_{2r} = v_{1v} = v_1 \sin \alpha \).

Kinetic energy at the outlet is \( \frac{v_2^2}{2} = \frac{v_1^2}{2} \sin^2 \alpha = \frac{v_1^2}{2} (1 - \cos^2 \alpha) \).

The only loss occurring within the flow is outlet kinetic energy. Therefore optimum operation corresponds to perpendicular outflow. Another loss is the downward head that is not used. The head used is somewhat lower than the geometrical head. As discussed in Chap. 1, some hydraulic turbines apply a draught tube to transform the downstream head into pressure decrease at the rotor outlet. The draught tube also functions as a diffuser and transforms a part of the outlet kinetic energy into pressure decrease at rotor outlet. Application of a draught tube is impossible with a Poncelet wheel, as the wheel runs in an open atmosphere.

References

Fundamentals of Turbomachines
Dick, E.
2015, XXVIII, 564 p. 337 illus., 16 illus. in color., Hardcover
ISBN: 978-94-017-9626-2