Lecture "Experimental Physics I"
(Prof. Dr. R. Seidel)

Lecture 22

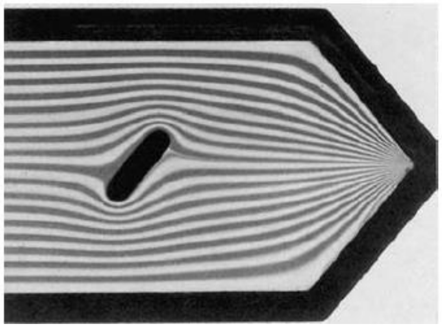
Hydrodynamics of “real” fluids

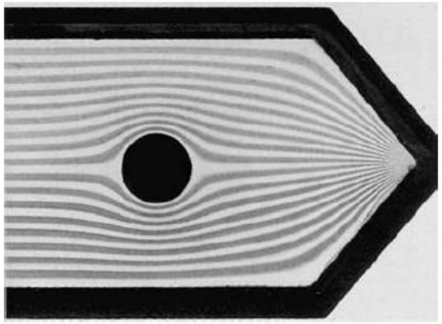
- Viscous flow & viscosity
- Navier-Stokes equation
- Stoke’s law
- Pressure driven flow
- Turbulent flow & applications
- Reynold’s number

1) Hydrodynamic equations with friction

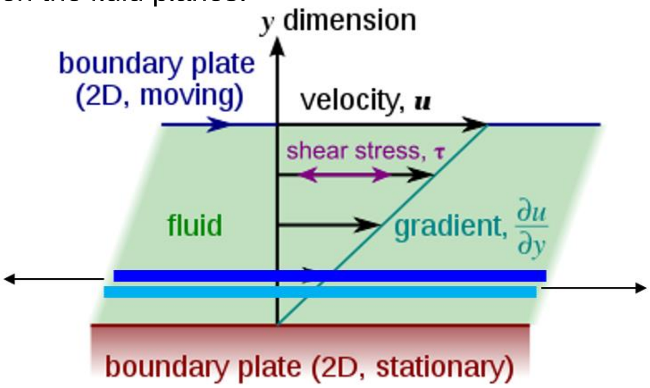
A) Internal friction of a laminar fluid flow

So far, we considered only ideal fluids in which no energy was dissipated (Euler equation, Bernoulli equation). Now let us get closer to reality by **considering friction within the fluid**. For this, we assume **laminar flow** (were stream lines do not cross each other, **see slides**)
In this case the friction forces are large over all possible inertial forces, i.e. forces that would accelerate the fluid elements. Friction forces occur due to the different velocities of neighboring stream lines (at walls but also within the bulk fluid).





A simple arrangement to understand internal friction is the **Couette flow**, where a **fluid gets sheared between two plates that move with velocity u relative to each other** (see below). For an ideal fluid no shear stress would arise, while **for a real fluid shear stress is observed** due to the friction between the fluid planes.



We typically **assume non-slip boundary conditions**, i.e. at each plate surface the relative flow velocity with respect to the plate surface is zero. Thus, the fluid has an absolute velocity of zero at the bottom and a velocity u at the top. Thus, we must have **a velocity gradient across the fluid in the vertical direction**.
Looking at the boundary between two fluid layers of infinitesimal height dy, we can write for their velocity difference:

$$du = \left(\frac{\partial u}{\partial y}\right) dy$$

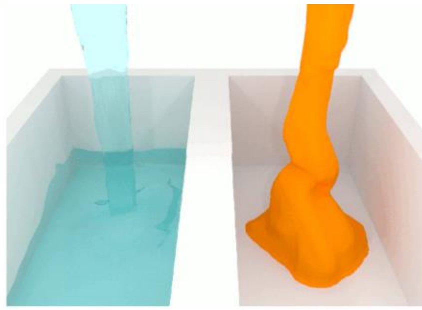
The **friction (shear) force between two layers is for many fluids proportional to the velocity gradient** such that it is given by:

$$|f| = \eta A \left|\frac{\partial u}{\partial y}\right|$$

The velocity gradient can also be interpreted as a shear rate (change in shear strain over time). A fluid for which **viscous stresses are proportional to the shear rate** is called **Newtonian fluid**.

The **proportionality constant η is called (dynamic) viscosity** with units of **[η] = Ns/m^2 = Pa s**.

The **viscosity** of a fluid is a **measure of its resistance to gradual deformation** by shear stress (or tensile stress). It can be imagined with the **informal concept of "thickness"**; for example, honey has a much higher viscosity than water, since the internal friction limits how rapidly a fluid spreads.



Now let us look at the **friction forces on top and the bottom of a single fluid layer** in the Couette flow:

They are given by the partial derivatives of the flow velocity along y as:

$$f(y_0) = -\eta A \left(\frac{\partial u}{\partial y}\right)_{y=y_0} \quad \text{and} \quad f(y_0 + \Delta y) = \eta A \left(\frac{\partial u}{\partial y}\right)_{y=y_0+\Delta y}$$

In steady state we have a constant velocity profile, such that the fluid layer is not accelerated. In this case the **sum of the friction forces must be zero** $0 = f(y_0) + f(y_0 + \Delta y)$, demanding that the derivatives are the same such that:

$$\eta A \left(\frac{\partial u}{\partial y}\right)_{y_0+\Delta y} = \eta A \left(\frac{\partial u}{\partial y}\right)_{y_0} = const.$$

This corresponds for our configuration to a constant gradient of the velocity profile or to a **linear velocity increase from the bottom to the top**. Thus, the same shear stress (force per area) acts between all fluid planes throughout the layer.

B) The molecular origin of viscosity

There are two important factors that cause viscosity:

1) **Molecular diffusion that transports momentum between layers of flow** provides the main friction component in gases. For this the theory predicts a viscosity increase with temperature, since the molecules increasingly diffuse between the layers of the medium. In this case viscous friction arises even without any molecular interactions!
2) **Forces between molecules,** which dominate in fluids. Since the weak bonds between molecules get easier broken at higher temperatures by thermal fluctuations, the fluid viscosities decrease with increasing temperature, which is also typically observed.

Fluids at 20°C

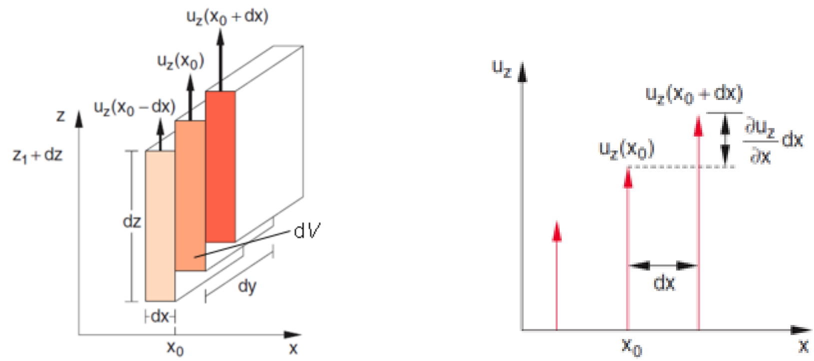
	η/(mPas)
water	1,002
Benzene	0,65
ethanol	1,20
glycerol	1480,0
mercury	1,55

Strong temperature dependence of viscosity

T/°C	$\eta(T)$/(mPas) water	$\eta(T)$/(mPas) glycerol
0	1,792	12100
+20	1,002	1480
+40	0,653	238
+60	0,466	81
+80	0,355	31,8
+100	0,282	14,8

C) General expression for internal friction on a volume element

After defining the friction force between two fluid layers, let us derive a general expression for the friction force at a given point of a flow field. For this, we determine the **effective force along z on a volume element** $dV = dx\, dy\, dz$ at position x_0 due to a **lateral gradient (along x) of the fluid velocity component u_z**:



We can use a Taylor expansion to express the velocity at position $x_0 + dx$ based on the velocity at position x_0:

$$u_z(x_0 + dx) = u_z(x_0) + \left(\frac{\partial u_z}{\partial x}\right)_{x_0} dx + \cdots$$

The same way we can express the velocity gradient at position $x_0 + dx$ based on the velocity gradient at position x_0:

$$\left(\frac{\partial u_z}{\partial x}\right)_{x_0+dx} = \left(\frac{\partial u_z}{\partial x}\right)_{x_0} + \left(\frac{\partial^2 u_z}{\partial x^2}\right)_{x_0} dx + \cdots$$

The effective force on dV is provided by the difference between the friction force on either side, since the faster right plane pulls dV forward while the slower left plane pulls dV backward (in case of a positive gradient as depicted on the right in the figure):

$$(\partial f)_z = df_z(x_0 + dx) - df_z(x_0)$$

Inserting the expression for the friction force at these two positions provides:

$$(\partial f)_z = \eta \underbrace{dy\, dz}_{dA} \left[\left(\frac{\partial u_z}{\partial x}\right)_{x_0+dx} - \left(\frac{\partial u_z}{\partial x}\right)_{x_0}\right]$$

Thus, the force is proportional to the change of the velocity gradient between x_0 and $x_0 + dx$. Inserting the Taylor approximation for the velocity gradient at $x_0 + dx$ provides

$$(\partial f)_z = \eta \underbrace{dy\, dz\, dx}_{dV} \left(\frac{\partial^2 u_z}{\partial x^2}\right)_{x_0} = \eta \left(\frac{\partial^2 u_z}{\partial x^2}\right)_{x_0} dV$$

i.e. the **friction force along z** on the volume element dV is **proportional to the 2nd derivative of u_z along x**. One can make the same argument for a velocity gradient along *y*. In case of a compressible medium one can also have a gradient of u_z along z and obtain a similar force component which acts normal onto the xy plane. This provides that the **total friction force along z** due is provided by a (non-constant) **gradient of u_z along all three directions:**

$$(\partial f)_z = \eta\, dV \left[\underbrace{\frac{\partial^2 u_z}{\partial x^2} + \frac{\partial^2 u_z}{\partial y^2}}_{\substack{\text{tangential}\\ \text{shear forces}}} + \underbrace{\frac{\partial^2 u_z}{\partial z^2}}_{\substack{\text{normal}\\ \text{force}}}\right] = \eta\, dV\, \Delta u_z$$

The normal force component is only non-zero for compressible media (only flow along z). The right side of the equation was obtained using the Laplace operator that is the scalar product between two Nabla-Operators:

$$\Delta = \mathrm{div}(\mathrm{grad}) = \vec{\nabla} \cdot \vec{\nabla} = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2}$$

Gradients of the x and y components of $\vec{u}$ provide corresponding friction force components along x and y. The general expression for the total internal friction onto a volume element dV is thus:

$$d\vec{f} = \eta\, dV\, \Delta\vec{u}$$

The total friction force in a given volume is finally provided by integrating the force over the volume:

$$\vec{f} = \int_V \eta\, \Delta\vec{u}\, dV$$

D) Navier-Stokes equation

Now we can add the expression for the friction as additional force term to the equation of motion of a small volume element derived before:

$$\Delta m \frac{d\vec{u}}{dt} = \underbrace{\rho\, \Delta V}_{\Delta m} \underbrace{\left[\frac{\partial\vec{u}}{\partial t} + (\vec{u}\cdot\vec{\nabla})\vec{u}\right]}_{\text{inertial term in Euler equation}} = \underbrace{-\vec{\nabla}p\, \Delta V}_{\substack{\text{force from}\\ \text{pressure difference}}} + \underbrace{\rho\, \Delta V \vec{g}}_{\text{gravity}} + \underbrace{\eta\, \Delta V\, \Delta\vec{u}}_{\text{internal friction}}$$

This finally provides the **Navier-Stokes equation** (in the form for incompressible media):

$$\rho\left[\frac{\partial}{\partial t} + (\vec{u}\cdot\vec{\nabla})\right]\vec{u} = -\vec{\nabla}p + \rho\vec{g} + \eta\, \Delta\vec{u}$$

This is the most central equation in fluid dynamics!

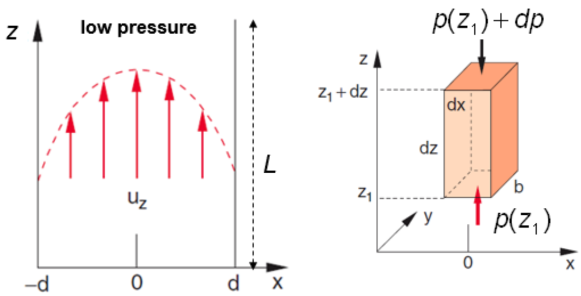
2) Pressure-driven flow

The Bernoulli equation explained us that the static pressure for an ideal fluid in a tube depends just on the local flow velocity if the tube is horizontally oriented.

Experiment: We measure the static pressure along a tube with constant cross section. We observe that there is a **decay of the pressure with length due to the friction opposing the movement of the fluid**. Thus, the work from a **pressure gradient is needed to overcome the internal friction of the fluid flow** in the construction.

A) Flow between two plates

To understand pressure driven flow through a constriction, we will calculate the pressure difference that is required to push a fluid with mean velocity u_z along z in between **two parallel infinite plates at $x = -d$ and $x = d$**. We will hereby **assume stationary laminar flow**, where inertial forces can be neglected.



At the plate surfaces the flow velocity is zero due to non-slip conditions. Due to the system symmetry we have a symmetric velocity profile with increasing velocity towards the center of the plates and thus a maximum velocity at x = 0.

For infinitely long walls we assume that there is only a negative **pressure gradient along $\boldsymbol{z}$**, since we would otherwise have lateral flows. This provides that we have a **constant pressure in horizontal direction at a given position $\boldsymbol{z}$**. We now look at the force on a ("deep") volume element $dV = dx\, b\, dz$ (see sketch). The force from the pressure gradient is given according to our findings in hydrostatics $d\vec{F} = -\vec{\nabla} p\, dV$ as:

$$dF_z = -\frac{\partial p}{\partial z}\, dV = -\underbrace{\frac{\partial p}{\partial z}\, dz}_{dp}\; b\, dx$$

where the negative sign ensures that the force acts towards the lower pressure (negative gradient). For stationary flow the pressure gradient force must be balanced by the opposing friction force such that we get:

$$0 = dF_z + df_r = -\frac{\partial p}{\partial z}\, dV + \eta\, dV \frac{\partial^2 u_z}{\partial x^2}$$

This equation corresponds to a simplified Navier-Stokes equation where the inertial and the gravity terms were omitted (see slides). Transformation gives:

$$\frac{\partial^2 u_z}{\partial x^2} = \frac{1}{\eta}\frac{\partial p}{\partial z}$$

Since the pressure (gradient) does not depend on *x* (same pressure along x) we obtain $u_z(x)$ by successive integration:

$$\frac{\partial u_z}{\partial x} = \frac{1}{\eta}\frac{\partial p}{\partial z} x + C_1$$

$$u_z = \frac{1}{2\eta}\frac{\partial p}{\partial z} x^2 + C_1 x + C_2$$

The integration constants are derived from the boundary conditions. From symmetry we must have a maximum of u_z at $x = 0$, such that

$$0 = \left(\frac{\partial u_z}{\partial x}\right)_{x=0} = C_1$$

From the non-slip condition at the plate surfaces we obtain:

$$0 = u_z(-d) = u_z(d) = \frac{1}{2\eta}\frac{\partial p}{\partial z} d^2 + C_2$$

Transformation and inserting finally provides a parabolic velocity profile:

$$u_z(x) = -\frac{1}{2\eta}\frac{\partial p}{\partial z}(d^2 - x^2) = -\frac{1}{2\eta}\frac{\Delta p}{\Delta L}(d^2 - x^2)$$

The **velocity is positive** in the equation, since **the pressure gradient is negative**. u_z is independent of $\boldsymbol{z}$ due to the **continuity equation**. Therefore $\partial p/\partial z = const$ and we can **replace** the **local pressure gradient by the macroscopic gradient $\boldsymbol{\Delta p/\Delta L}$**.

The fluid volume that passes through the cross section area $b\, dx$ within time t is:

$$dV = \underbrace{u_z t}_{\Delta z}\; b\, dx$$

The total volume that passes through the cross-section is obtained by integration over the whole velocity profile:

$$V = \int_{-d}^{d} u_z t\, b\, dx = -\frac{b}{2\eta}\frac{\Delta p}{\Delta L}\underbrace{\left(d^2 x - \frac{x^3}{3}\right)\Bigg|_{-d}^{d}}_{\frac{2}{3}d^3 - \left(-\frac{2}{3}d^3\right)}\; t = -\frac{2b}{3\eta}\frac{\Delta p}{\Delta L} d^3\, t$$

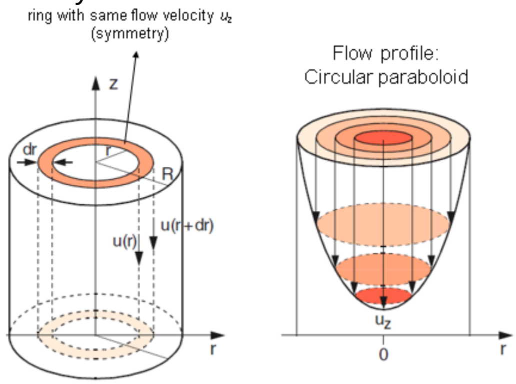
The volume current is thus proportional to the pressure difference, inversely proportional to L and proportional to the **third power of the plate distance**.

B) Flow through a tube

The same approach can be taken for calculating the flow through a tube/pipe. Here one just has to use cylindrical (polar) coordinates and consider the friction between thin hollow cylinders of equal velocity. (**see slides**)
We can use the same differential equation as before for a small segment of the hollow cylinder:

$$-\frac{\partial p}{\partial z}dV = -\eta\, dV\, \Delta_r u_z = -\eta\, dV \underbrace{\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right)}_{\substack{\text{Laplacian for r}\\ \text{in cyl. coordinates}}}$$

The term on the right side is the Laplacian in polar coordinates that should be used if there is only a radial dependence of the velocity



With this one obtains a quadratic relationship for u_z as function of the radius, very similar as before. Due to the cylindrical symmetry the obtained flow profile is a circular paraboloid:

$$u_z(x) = \frac{1}{4\eta}\frac{\Delta p}{L}(R^2 - r^2)$$

The volume current is in this case proportional to the 4th power of the tube radius:

$$\frac{V}{t} = \frac{\pi}{8\eta}\frac{\Delta p}{L}R^4$$

which is known as the **law of Hagen-Poiseuille**.

Experiments:

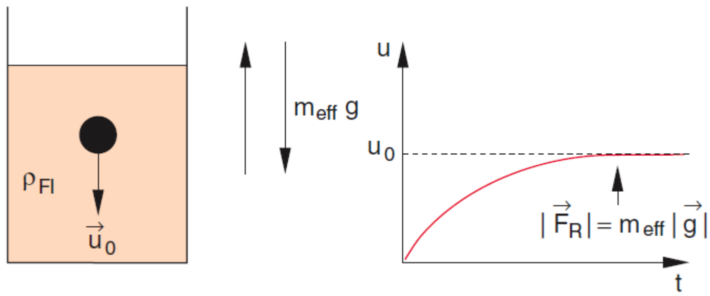
- Paraboloid-shaped velocity profile through tube
- Scaling of volume current with R^4

3) Stoke’s law

Another important occurrence of viscous friction forces are moving objects in viscous fluids.

Experiment: We look at the “free” fall of a sphere in a viscous solution. One observes that the **sphere reaches after a while a constant velocity**, which is different to the constant free fall acceleration in vacuum!!!

From the derivations above we see that the flow velocity is proportional to the pressure gradient that balances the friction. Thus, the friction forces on the moving sphere should also increase with its velocity until they balance the gravity force and the acceleration stops.



One can show with elaborate calculations that the friction force on a sphere moving through a viscous solution increases proportionally with its velocity. It is given by Stoke's law:

$$\vec{f}_r = -\underbrace{6\pi\eta R}_{\text{drag coefficient } \gamma}\ \vec{u}_0$$

where the term before the velocity is the **drag coefficient $\boldsymbol{\gamma}$ of the object,** $[\gamma]$ = Ns/m. In steady state when the final velocity is reached, the effective gravity force of the sphere is compensated by the friction:

$$\vec{F}_g^* + \vec{f}_r = 0$$

Hereby the effective gravity force is reduced by the buoyancy such that we can write:

$$-\left(\rho_{sphere} - \rho_{fuid}\right)\frac{4}{3}\pi R^3 g + 6\pi\eta R\, u_0 = 0$$

This provides:

$$u_0 = \left(\rho_{sphere} - \rho_{fuid}\right)\frac{2}{9}\frac{g}{\eta}R^2$$

When knowing the densities one can use such an experimental setting to determine the viscosity of a fluid.

The Stokes friction force is also observed **on a resting sphere in a fluid that moves with (bulk) velocity $\boldsymbol{u}$**. For a resting sphere in a moving fluid the same formula applies, since just the relative motion between sphere and fluid is important!

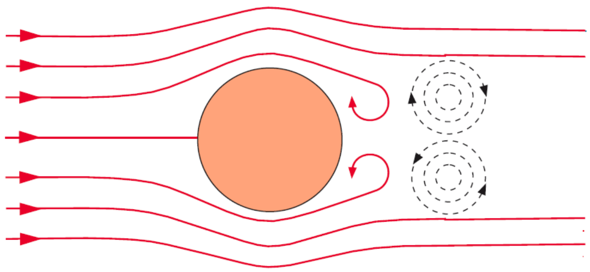
4) Turbulent flow: Vortex(es)

A) Properties and generation

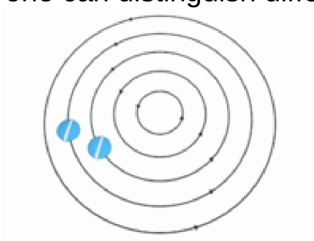
So far, we considered only laminar flow, where one typically looks at stationary flow and often neglects inertia. Now let us look at **turbulent flow, which can be quite chaotic** and disordered.

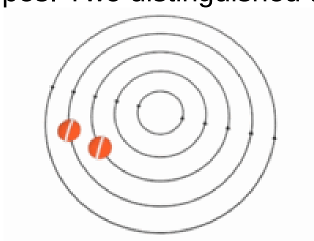
Video: A line of ink in a viscous solution illustrates nicely the transition from laminar to turbulent flow.

Within turbulent flow, **vortices are more ordered and reoccurring features that can absorb large amount of the kinetic energy of the flow**. Vortices form typically behind objects at velocities where the inertia of the fluid elements matter. They are a result of the interplay between inertial forces and friction forces at the boundary of an object. A well-known example is the plug-hole vortex (**see slides**)



More specifically one can distinguish different vortex types. Two distinguished types are:

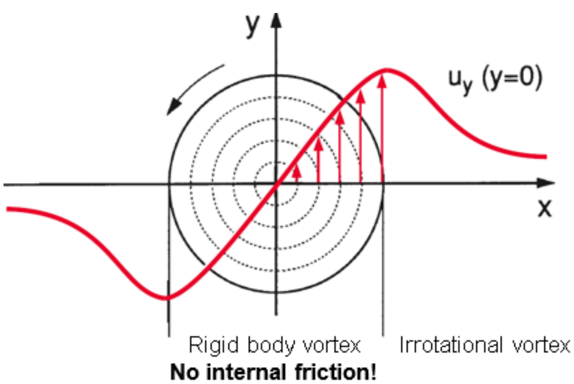




the **rigid body vortex** and the **irrotational vortex.**

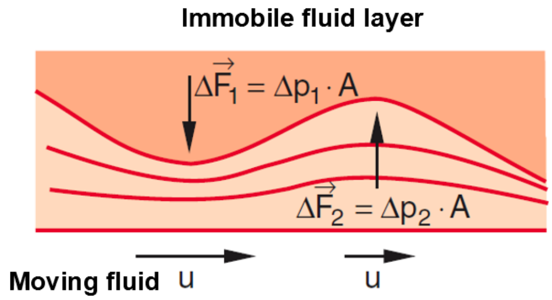
The **rigid body vortex** is a vortex were the fluid elements possess the same angular velocity (i.e. the tangential velocity increases proportionally with the radius) such that they rotate once around their center per vortex rotation.
The irrotational vortex is a vortex where the tangential velocity of fluid elements decreases with 1/r, such that the fluid elements do not rotate around their centers

Real vortices are more complex and can often be described as **hybrids of a rigid body vortex in the core of the vortex and an irrotational vortex in the periphery**



The tangential velocity thus increases linearly up to the core boundary and decays further outwards. As for a rigid body, there is **no friction between the fluid elements in the vortex core** but only in the periphery. Vortices form therefore quite stable reservoirs in which kinetic energy becomes stored ("trapped"). This is e.g. problematic for vortices formed behind aircrafts - **called wake vortices or wake turbulence** – since they can persist over length scales of kilometers, such that corresponding safety distances must be maintained.

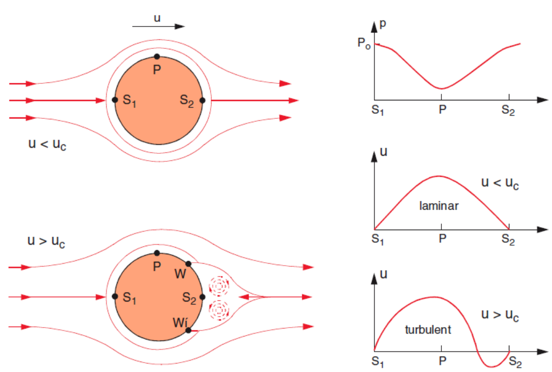
Vortices originate at interface instabilities (e.g. small protrusions) between moving and more immobile fluid layers:



At such a protrusion the fluid flows faster. This causes a lower pressure than in the region behind. This local pressure difference is thus causing a torque, making the instability even larger at sufficiently high flow velocities.

This occurs also behind an object. Here we have slow fluid layers near the object's surface and faster fluid layers further away from the object. We have different scenarios depending on the flow type (see figure below):

- Laminar flow: There is a lower pressure around the midpoint of object (Bernoulli equation) where the fluid velocity is the highest. There are only positive flow positive velocities that approach zero at the stagnation points S1 and S2.
- Turbulent flow: Due to friction, kinetic energy is lost and the flow velocity along the surface becomes zero already before S_2 (at W in the figure). The pressure difference from S2 towards W (where we have a flow velocity in between) causes now a backdriving flow with negative velocity from S2. At W we have an increased static pressure and thus, a flow normal to and away from the surface such that a flow direction reversal takes place. Together this causes the formation of vortices. These vertices are called eddies, if they are behind the object.



Vortices are, however, not stably bound but typically detach from the object and move like independent objects for which conservation laws like momentum conservation holds. Vortex detachment can e.g. be seen when pulling a paddle fast enough through the water.

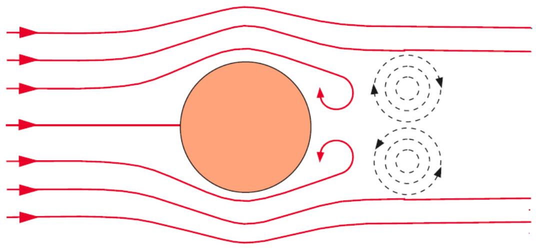
Experiment: Vortex "canon" that creates toroidal vertices that blow out a candle

Slide: Karman vortex street with an alternating vortex detachment at either object side

Movie: Wake vortices at aircraft wings

B) Drag in case of turbulent flow

Why do we care about vortices? Their continuous generation costs energy that is "taken away" from the object that generates the flow (e.g. the aircraft, the biker, the ship). Thus, there must be an additional force that the object has to generate in order to "power" the vortices.



In simplified form this can be understood by considering the increased fluid velocity behind the object due to the presence of the vortices. According to Bernoulli this causes a reduced static pressure, such that in addition to viscous friction we have:

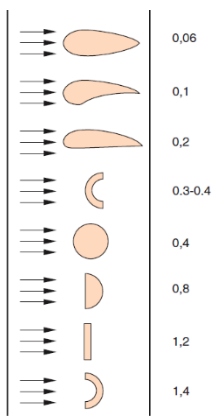
$$\Delta p \propto c_w \frac{\rho}{2} u^2$$

with the **additional turbulent drag being quadratically dependent on the velocity**. Thus, the force becomes:

$$F_{drag} = c_w \frac{\rho}{2} u^2 A$$

with $\boldsymbol{c_w}$ being **the drag coefficient for turbulent flow.** c_w is a **unit less number** and should not be confused with drag coefficient of laminar flow. Turbulent drag forces have main differences compared to laminar flow. The turbulent drag forces scale quadratically rather than linearly with the velocity. They also scale with the cross-sectional area of the object, i.e. with the squared radius of the object, rather than the radius itself. Thus, minimizing the cross-section to reduce the drag is more advantageous for turbulent compared to laminar flow (and exactly the opposite what car buyers do when purchasing an SUV ☹).

Furthermore, the drag is heavily influenced by the shape of the object as we know from everyday experience (compare open plastic bag with ball of same cross-section). The shape dependence is described by the c_w-value. The following table provides an overview over the large spread of the c_w-value for different shapes.



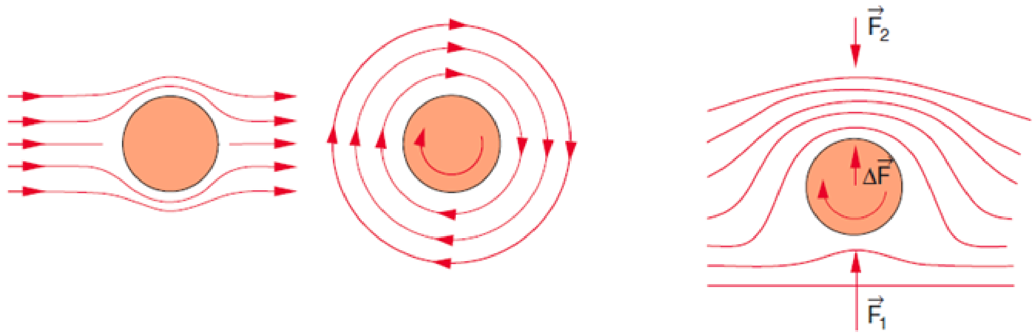
This quantity is the one that e.g. car manufacturers try to optimize for a given cross sectional area to minimize drag forces. Optimizing the c_w value is done by preventing vortices to form. The best way is to shape the objects similar to the occurring streamlines. Streamline shaped objects have therefore much lower c_w then typical geometric shapes (e.g. discs, spheres, etc.)

Experiment: Qualitatively assessing the drag forces onto differently shapes objects in air flow
Movie: Steady state fall velocity can be modified by changing the c_w ("Moonraker").

C) Lift force in aerodynamics

For understanding the lift of aircrafts, one also has to consider vortices. It cannot be done by applying the Bernoulli equation alone, otherwise an aircraft could not fly upside down. The latter can only be explained by the consideration of turbulences.

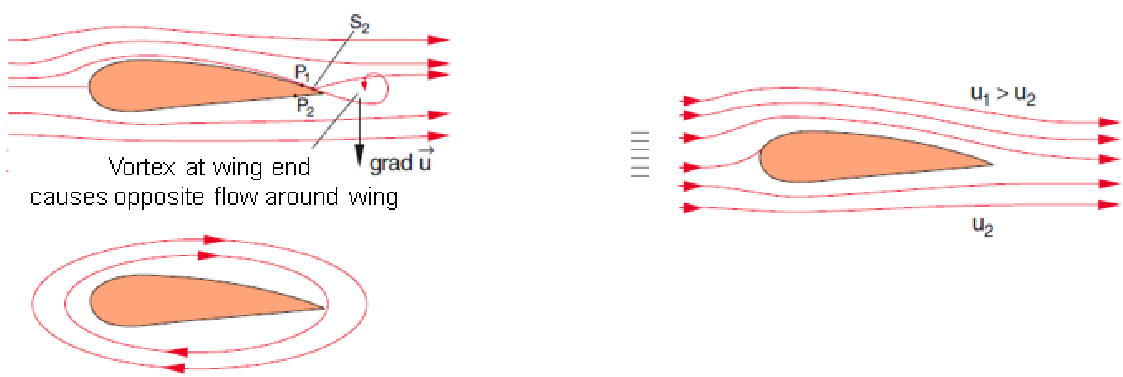
A main contribution is coming from a flow around the wing, if subtracting the air velocity. To understand this better, we look at the **Magnus effect**, which describes that a rotating cylinder in a flow experiences a force perpendicular to the flow direction:



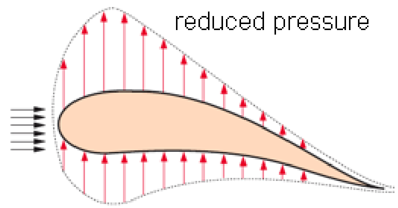
In absence of external flow, the friction forces at the rotating cylinder surface make the medium moving on one side in the flow direction and on the other side against the flow direction.

Superposition of the rotational flow with the external linear flow causes therefore a higher velocity on one and a lower velocity on the opposing side of the cylinder. This causes an asymmetric flow profile. According to Bernoulli we have thus a reduced static pressure on the high and an increase static pressure at the low velocity side and thus a corresponding force perpendicular to the flow direction.

A similar concept explains the **lift at the aircraft wing**:



A vortex at the end of the wing is upward oriented due to the longer way of the air along the upper part of the wing which due to friction leads to a lower velocity. Angular momentum conservation requires an air rotation in the opposite direction to compensate the angular momentum of the vortex at the wing end. This opposing air rotation generates a flow around the aircraft wing. Overlaying this with the external linear airflow provides a reduced velocity at the bottom and an increased velocity at the upper side of the wing. This in turn causes a pressure difference and thus the lift on the wing. The pilot can influence the vortex and therefore the lift using its elevators (blades at the wings).



Furthermore, the deflection of the air behind the wing plays an important role. For an upward tilted wing the air is deflected downwards, such that momentum conservation provides an upward force. This is intuitive, since even a board at a given angle experiences a lift. Overall the understanding of lift forces at air crafts is a quite complex subject.

5) Reynolds number

A) Deriving the Reynolds number

Let us in the following see, if we can find a parameter that tells us whether we can expect a turbulent or a non-turbulent flow. More practically this addresses the following two points: How do the different types of flow change when altering the length of an object? For example, if we want to use a small model to mimic the flow around a ship or an aircraft it would be useful to know which flow velocity we have to choose to get the same flow profile?.

To address these questions, we have to understand **how we need to scale the (bulk) flow velocity** (and/or time) **when changing the size of an object** (e.g. using a smaller model)**.** For example, imagine we shrink the size of an object by a factor of 2. To obtain a sensible downscaling from the physics point of view, it would be important that still the same flow type/pattern (incl. turbulences) would be obtained. How do we have to scale then the bulk flow velocity to achieve this? Do we have to reduce it also by a factor of 2?

To see how an **appropriate scaling** can be obtained we introduce the following scaling factors:

- We take the characteristic length scale L of the system (e.g. the diameter of the sphere/ship/aircraft) and its characteristic velocity U (mean fluid velocity, bulk velocity, velocity of the object)
- We rescale position and velocity with respect to these characteristic values, to get dimensionless parameters:
$$x' = x/L$$
$$\vec{u}' = \vec{u}/U$$
- The rescaling of length and velocity suggests a corresponding rescaling of the time using the characteristic time L/U, which the flow needs to pass the characteristic length scale L:
$$t' = t/\underbrace{(L/U)}_{\text{t to pass}}$$
- Rescaling of positions and time leads correspondingly to a rescaling of the position- and time-derivatives:
$$\vec{\nabla} = \frac{\vec{\nabla}'}{L}$$
$$\frac{\partial}{\partial t} = \frac{\partial}{\partial t'}\frac{U}{L}$$
- Motivated by the Bernoulli equation, the pressure is rescaled by
$$p' = p/(U^2\rho)$$
Though we note that for the pressure scaling there is no natural reference scale.

The chosen rescaling is somewhat arbitrary, such that one could also use different possibilities. However, by inserting our rescaling into the Navier-Stokes Equation we directly see its value. The Navier-Stokes equation was in its original form:

$$\rho\left[\frac{\partial}{\partial t}+(\vec{u}\cdot\vec{\nabla})\right]\vec{u}=-\vec{\nabla}p+\eta\,\Delta\vec{u}$$

Inserting gives:

$$\rho\left[\frac{\partial}{\partial t'}\frac{U}{L}+U\vec{u}'\,\frac{\vec{\nabla}'}{L}\right]\vec{u}U=-\frac{\vec{\nabla}'}{L}p'\,U^2\rho+\eta\cdot\frac{U}{L^2}\Delta\vec{u}'\Bigg|:\frac{\rho U^2}{L}$$

Division by $\rho U^2/L$ provides the non-dimensional Navier-Stokes equation:

$$\left[\frac{\partial}{\partial t'}+(\vec{u}'\cdot\vec{\nabla}')\right]\vec{u}'=-\vec{\nabla}p'+\frac{1}{Re}\Delta\vec{u}'$$

With $\boldsymbol{Re}$ being the **unitless Reynolds number**:

$$Re=\frac{\rho}{\eta}U\,L$$

The non-dimensional Navier-Stokes equation allows us to draw an **important conclusion:**

> Independent of how large the real values of L and U are**,** one obtains **the same solution for the flow for miniature model and real object, if the Reynolds number is the same!**

This holds even in the case of turbulence. After solving the non-dimensional equation, one just needs to back-scale length, velocity, time and pressure to get results in real physical dimensions. Based on the derived equation, the scaling of the velocity is not intuitive:

Example: A length reduction by a factor of 2 requires a velocity increase by 2 to keep the Reynolds number constant. The flow passes then the miniature object 4 times faster. This is actually the opposite of what we initially hoped, were we hypothesized a 2-fold velocity reduction upon a 2-fold length reduction. Thus, **downscaling size and velocity changes the flow pattern**.

Additionally, the **Reynolds number tells how laminar/turbulent a flow is**. We had before:

- In laminar flow the friction term becomes large and the inertial term can be neglected.
- In turbulent flow the friction term can be (almost) neglected and the inertial term dominates.

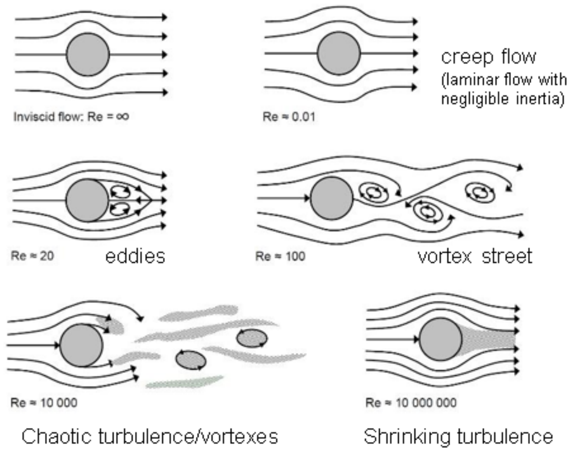
Both contributions enter also the Reynolds number. This can be seen by multiplying numerator and denominator of the expression with UL^2:

$$Re=\frac{\rho}{\eta}U\,L=\frac{\rho\,U^2L^3}{\eta\,UL^2}\approx\frac{m\,U^2}{F_{drag}\,L}=\frac{2E_{kin}}{W_{drag}}$$

The further transformation used considered that $\rho\,L^3$ is approximately the mass of the fluid in the flow profile around the object and ηUL is approximately the stokes friction term. With this we get that the Reynolds number corresponds approximately to the ratio between the kinetic energy of the fluid and the friction work done in the fluid. This means:

- At **low** Reynolds the **friction term becomes large** and we have **laminar flow**.
- At **high** Reynolds numbers the **inertial term dominates** and we have **turbulent flow**.

On a disk (cylinder in 3D) one gets the following flow behavior for the given Reynolds numbers. The actual flow velocities and disk radii do not matter, but rather the Reynolds number calculated from both.



At **very high Reynolds-Numbers** (large object, very high speeds) the **flow approaches an inviscid turbulent flow**. **Thus, turbulence** is a consequence of the **simultaneous acting friction and inertial forces**

B) Life at low Reynolds numbers

Microscopic objects (bacteria, proteins) will practically **always experience $\boldsymbol{Re} \ll \mathbf{1}$** due to their small length scale that enters the Reynolds number at the relevant flow velocities. Microscopic objects thus experience only **laminar flow conditions.**
At laminar flow conditions friction dominates over any inertia due to the viscosity of the fluid. The left-hand side of the Navier-Stokes equation can then be neglected and one gets the simple form:

$$0 = -\vec{\nabla} p + \eta \, \Delta \vec{u}$$

A special property of this equation is that it allows time-reversed flows as solution, if the external forces are also reversed. This means, if the movement of an object is subsequently done in a reverse manner, we get the original state of liquid and object back.

Movie: Reversal of dye droplet shearing in a viscous fluid

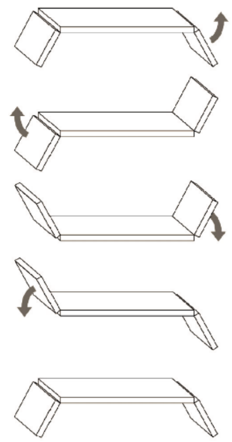
Time reversibility **requires a complete different way of swimming** compared to humans who use vortices and thus momentum transfer to propel in water.
Edward Mills Purcell laid the fundations of hydrodnymics at the microscale in his publication" **Life at Low Reynolds Number**" (1977). He formulated the so-called **Scallop theorem**:

> To achieve propulsion at low Reynolds number in Newtonian fluids, a swimmer must deform in a way that is not invariant under time-reversal.

To understand this, one has to consider that a **real Scallop uses turbulent flow to propel itself**. It opens slowly and closes rapidly. Thus, the **swimming exploits the hydrodynamic recoil** from accelerating the ejected water masses.

see movie: Swimming scallops

A **microscopic scallop** would experience only **low Reynolds numbers and thus laminar flow** conditions. The **two periodic beats** of the scallop (opening + closing) would be **reciprocal to each other**, i.e. the **closing is time reversed to the opening**. Thus, one would obtain the movement during closure by playing the movie taken from the opening backwards. Similar to the droplet shearing experiment, **two time-reversed beats do in this case not provide forward motion but rather the original state**, such that the micro-scallop would not move forward.
Swimmers in the low Reynolds number world must break time-reversal. The simplest scheme involves two joints is depicted below:



This principle is e.g. used by beating sperm tails. An alternative would be a continuous cork-screw motion.

Lecture 22: Experiments

1) Static pressure decay along tube with constant cross section.
2) Hagen-Poiseuille: paraboloid-shaped velocity profile through tube
3) Hagen-Poiseuille: Scaling of volume current with R^4
4) Stoke's law: "free" fall of a sphere in a viscous solution, where the sphere reaches a constant sink velocity after a period of decreasing acceleration.
5) Movie: Line of ink from laminar to turbulent
6) Vortex canon that blows out a candle
7) Magnus effect using a spinning falling cylinder, movie of falling ball from reservoir dam (see slides)
8) c_w-value of different objects in air flow qualitatively
9) Reversal of dye droplet shearing in a viscous fluid