CFD Examination questions

1. Explain why CFD is both a powerful and a dangerous tool.

CFD is “solving fluid flow problems numerically” (with the computer)

CFD is becoming increasingly important in research and design, in consultancy and industry, and in all domains of engineering, continues to grow. This is caused by (1) the increasing awareness of the power of this approach; (2) the increase in computational resources; and (3) the availability of increasingly user-friendly and powerful CFD software. At the same time, these reasons are also the reasons why CFD is often criticized.

Due to the availability of user-friendly software and powerful computers, it has become possible, even for non-skilled users, to generate results with CFD, which are not necessarily accurate, and which can even be completely incorrect. CFD results are often presented as colored contours and/or vector plots, which might seem impressive and convincing, even if the results are incorrect or unrealistic. This fact has lead critics to refer to the acronym CFD with cynical terms such as “Colorful Fluid Dynamics”, “Color For Designers”, etc.

Computational Fluid Dynamics is a tool that allows us to solve flow problems that do not have known analytical solutions and cannot be solved in any other way.

Obtaining accurate and reliable results with CFD is difficult and requires a considerable amount of knowledge and experience. Because many users of CFD do not have this basic knowledge, large errors can be made, and as a result many professionals do not trust CFD results. While CFD can certainly provide incorrect results and can be a dangerous tool, it can also be a very powerful tool, if used in the correct way. The safest approach in CFD is always to question the accuracy and reliability of the results, whether they are your own or somebody else’s. There is no assumption of innocence in CFD: **CFD results are wrong, until proven otherwise**.

The safest approach is to **always question the accuracy of CFD results**, whether they are your own or somebody else’s.

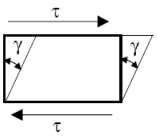
2. Write the Greek alphabet (in lower-case letters) and give the English names of the letters.

Case		English Name
Upper	Lower	
Α	α	alpha
Β	β	beta
Γ	γ	gamma
Δ	δ	delta
Ε	ϵ	epsilon
Ζ	ζ	zeta
Η	η	eta
Θ	θ	theta
Ι	ι	iota
Κ	κ	kappa
Λ	λ	lambda
Μ	μ	mu

Case		English Name
Upper	Lower	
Ν	ν	nu
Ξ	ξ	xi
Ο	ο	omicron
Π	π	pi
Ρ	ρ	rho
Σ	ς	sigma
Τ	τ	tau
Υ	υ	upsilon
Φ	ϕ	phi
Χ	χ	chi
Ψ	ψ	psi
Ω	ω	omega

3. What fluid mechanics variables are indicated with the Greek letters: μ, ν, κ, η, ρ and ε? Provide a brief definition of these variables or briefly explain their meaning.

- **μ (mu) (Dynamic molecular viscosity)** also *absolute viscosity*, the more usual one (kg/ms or Pa.s, Poise, P).



$$\tau = \mu \frac{\partial \gamma}{\partial t} \quad \text{or} \quad \tau = \rho \nu \frac{\partial \gamma}{\partial t}$$

μ = dynamic molecular viscosity (kg/ms or Pa.s)

ν = kinematic molecular viscosity (m^2/s)

Viscosity is a material property, unique to fluids, that measures the fluid's resistance to flow. Though a property of the fluid, its effect is understood only when the fluid is in motion. When different elements move with different velocities, each element tries to drag its neighboring elements along with it. Thus, shear stress occurs between fluid elements of different velocities.

If a fluid with a viscosity of one Pa·s is placed between two plates, and one plate is pushed sideways with a shear stress of one pascal, it moves a distance equal to the thickness of the layer between the plates in one second.

Dynamic viscosity of air **increases** with increasing temperature.
Dynamic viscosity of water **decreases** with increasing temperature.

- **ν (nu) (Kinematic molecular viscosity)** is the *dynamic viscosity* divided by the density (typical units m^2/s, Stokes, St).

In many situations, we are concerned with the ratio of the inertial force to the viscous force (i.e. the Reynolds number, Re = VD / ν), the latter characterized by the fluid density ρ. This ratio is characterized by the kinematic viscosity defined as follows:

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

High viscosity - high resistance to deformation (e.g. stirring in syrup)
Low viscosity - low resistance to deformation (e.g. stirring in water)
Viscosity is a measure of the resistance of a fluid to deform under shear stress.
Viscosity depends on temperature (and pressure)!

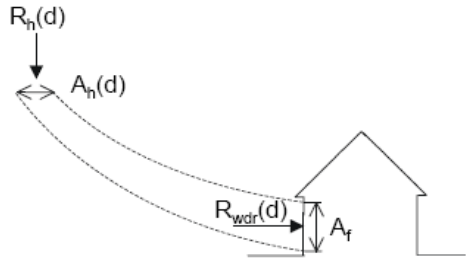
- **κ (Von Karman constant)**

The Von Karman constant has been found to be independent of the physical nature of the surface. Slightly different values for ε have been suggested by different authors (values between 0.38 and 0.42). The Von Karman constant is a dimensionless constant describing the logarithmic velocity profile of a turbulent fluid flow near a boundary with a no-slip condition. The equation for such boundary layer flow profiles is:

- **η (catch ratio)**

Used at wind-driven rain. $\eta = R_{wdr} / R_h = A_h / A_f$. Specific WDR intensity on the building and the specific unobstructed horizontal rainfall intensity, integrated over all raindrop diameters. The catch ratio is influenced by <u>6 parameters</u> :

geometry building	wind direction
position on façade	horizontal rainfall intensity
wind speed	horizontal raindrop size



- **ρ (rho) (Density)**

Fluid density is mass per unit volume

$$\rho = \frac{m}{V}$$

The mass density or density of a material is defined as its mass per unit volume. Density is mass divided by volume. The symbol most often used for density is ρ (the Greek letter rho). Different materials usually have different densities, so density is an important concept regarding buoyancy, purity and packaging.

The mass density of a material varies with temperature and pressure. (The variance is typically small for solids and liquids and much greater for gasses.) Increasing the pressure on an object decreases the volume of the object and therefore increases its density. Increasing the temperature of a substance (with some exceptions) decreases its density by increasing the volume of that substance. In most materials, heating the bottom of a fluid results in convection of the heat from bottom to top of the fluid due to the decrease of the density of the heated fluid. This causes it to rise relative to more dense unheated material.

- **ε (the turbulence dissipation rate)** (m^2/s^3)

The turbulence dissipation rate is the rate of conversion of turbulence into heat by molecular viscosity.

Turbulence dissipation is greatest for the smallest-size eddies (on the order of millimeters in diameter), but turbulence is usually produced as larger eddies roughly the size of the atmospheric boundary layer (on the order of hundreds of meters).

$$\varepsilon = \frac{\boldsymbol{u^3(y + y^0)}}{\boldsymbol{\kappa}}$$

4. Give and explain the three subdivisions of building physics. Also give practical examples of how CFD can be useful in each of these subdivisions.

The physics of the built environment to ensure a healthy, comfortable and durable outdoor and indoor environment of buildings. This must include economic, environmental and energy considerations are taken along. The physics of the built environment is also a study of the relevant physical processes in and around buildings as heat transfer, air, moisture and mold, light and sound. The physics of the built environment can be divided into three divisions.

Physics of the indoor
In the physics of the interior should especially be given to the ventilation and thermal comfort of a room. The ventilation of a space is very important to the health of people in space, but also for preventing mold in one room. It should take into account air pollution polled. This air must be purified as much as possible. The ventilation has a direct link with the thermal comfort of a room. The more that ventilation (cold air from outside to inside out), the colder it in a room and often a decreasing thermal comfort in space. In order to determine progress can be made very good use of CFD. This allows the air flow in a given space.

Physics of the building envelope
The shell can experience a number of processes during the day. The shell is exposed to bad weather such as wind, rain, sun, etc. These all affect the building envelope. One should think of changing surface temperatures and moisture transport through the skin. But energy loss and air infiltration plays a major role. All these processes can be simulated using CFD. In this way can be pre-determined or moisture problems may arise from the moisture transport of a wall to simulate. But also can determine whether and how much heat is transported by a wall.

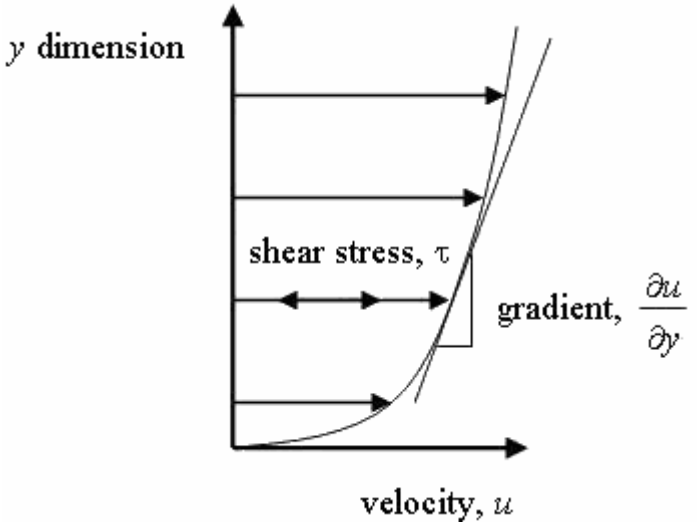
Physics of the environment
Outside a building physics plays a role. It should for example think of wind comfort. The placement of buildings relative to each other plays a key role in the behavior of wind. For simple buildings can still be analytically to determine the course winds around the building will be. For more complex buildings or multiple buildings together is more difficult to impossible. For these cases it is very convenient to use CFD. This allows large wind problems, including problems with the wind for pedestrians, be prevented. Another aspect that follows is the spread of polluted air. A building can be polluted air that flows back into a certain direction. Using CFD is easy to determine where the polluted air is flowing and whether this might be in a different building to enter. To avoid a CFD simulation is highly desirable, because this is not always amenable to analytical determination.

5. Explain the terms “molecular viscosity” and “turbulent viscosity”. How are they defined? What process causes viscosity? What is the difference between both viscosities?

Viscosity is a material property, unique to fluids, that measures the fluid's resistance to flow. Though a property of the fluid, its effect is understood only when the fluid is in motion. When different elements move with different velocities, each element tries to drag its neighboring

elements along with it. Thus, shear stress occurs between fluid elements of different velocities.

High viscosity means high resistance to deformation (e.g. stirring in sirup).
Low viscosity means low resistance to deformation (e.g. stirring in water).



Viscosity depends on temperature and pressure
Dynamic viscosity of air **increases** with increasing temperature.
Dynamic viscosity of water **decreases** with increasing temperature.

Molecular viscosity
Describes a fluids molecular transport of momentum. **$\mu = \rho \cdot v$ [kg/ms or Pa.s]**
ρ = fluid density (kg/m³)
v= kinematic molecular viscosity (m²/s).

Turbulent viscosity
Besides molecular transport of momentum turbulent viscosity takes into account the turbulent transport by regarding small scales turbulent eddies as “macro-molecules”.
The turbulent viscosity is not an isotropic scalar quantity as assumed in the k-ε model. An advantage is the low computational costs due to the use of the turbulent viscosity.

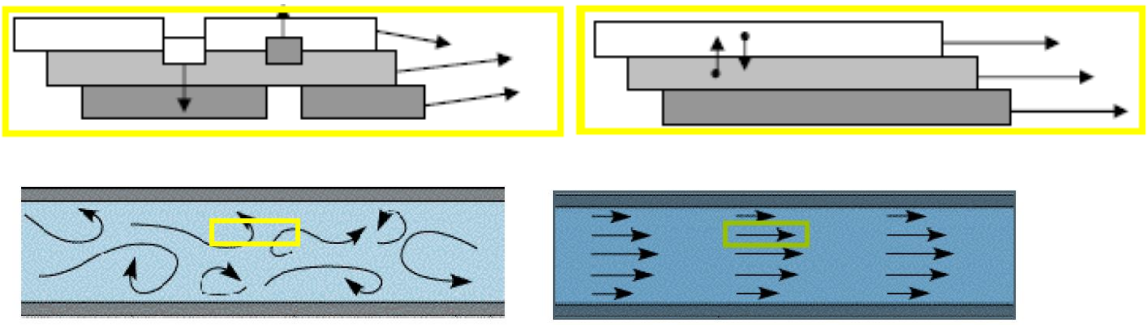
$\mu_t = \rho \cdot C_\mu \cdot (k^2/\varepsilon)$ [kg/ms]

ρ = fluid density (kg/m³)
Cμ = parameter in the k-ε model (-)
K = turbulent kinetic energy (m²/s²)
ε = turbulence dissipation rate (m²/s³)

What process causes viscosity?
Viscosity originates from the exchange of momentum between fluid layers caused by exchange of molecules. The momentum exchange creates sheer stresses between fluid layers. Viscosity is a measure of the resistance of a fluid to deform under shear stress; a high viscosity means a high resistance to deformation. Viscosity depends on temperature and pressure.

What is the difference between both viscosities
The difference between both viscosities is that molecular viscosity is present in laminar and turbulent flows, <u>while turbulent viscosity is only present in turbulent flows.</u>

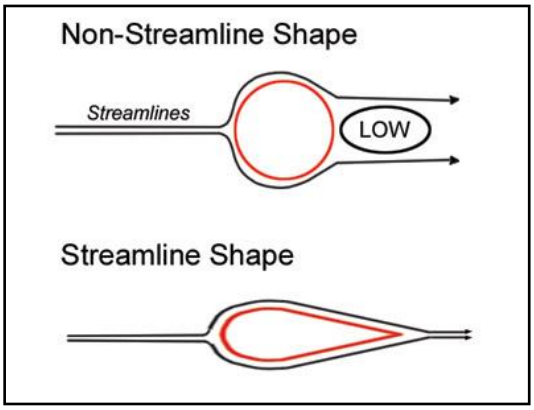
A laminar flow is a smooth flow where only molecules are exchanged between the different fluid layers (exchange of a quantity of movement) and a turbulent flow is a chaotic flow where fluid particles (bunches of molecules) are exchanged between different fluid layers (exchange of a quantity of movement), due to the whirly movements in turbulent flows.
Het verschil tussen beide viscositeiten is dat: moleculaire viscositeit aanwezig is in laminaire en turbulente stromingen, terwijl de turbulente viscositeit is alleen aanwezig in turbulente stromingen.



Viscosity originates from the exchange of momentum between fluid layers caused by exchange of molecules.

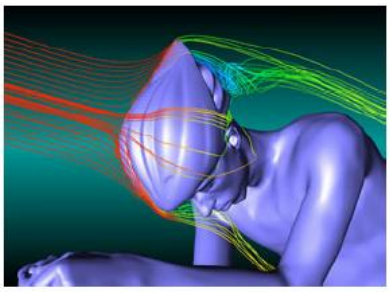
6. Explain the terms homogeneity, isotropy, streamline, streamtube and provide drawings.
What is the opposite of homogeneity and isotropy?

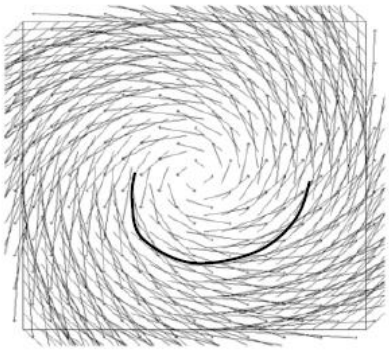
A streamline is a line that, at some point in time, tangential (tangent) is the vector of the velocity at all points. There is thus a vector in each point of the speed (vector length) and direction of a particle in that paragraph indicates. He therefore gives the direction of each point in a flow. 2 Streamline can never cross. This would mean two different velocity vectors in a point, which is impossible. The boundaries of a stream streamline. In a snapshot of a flowing material: one line, in which each point the tangent line coincides with the velocity vector of the volume element in this section



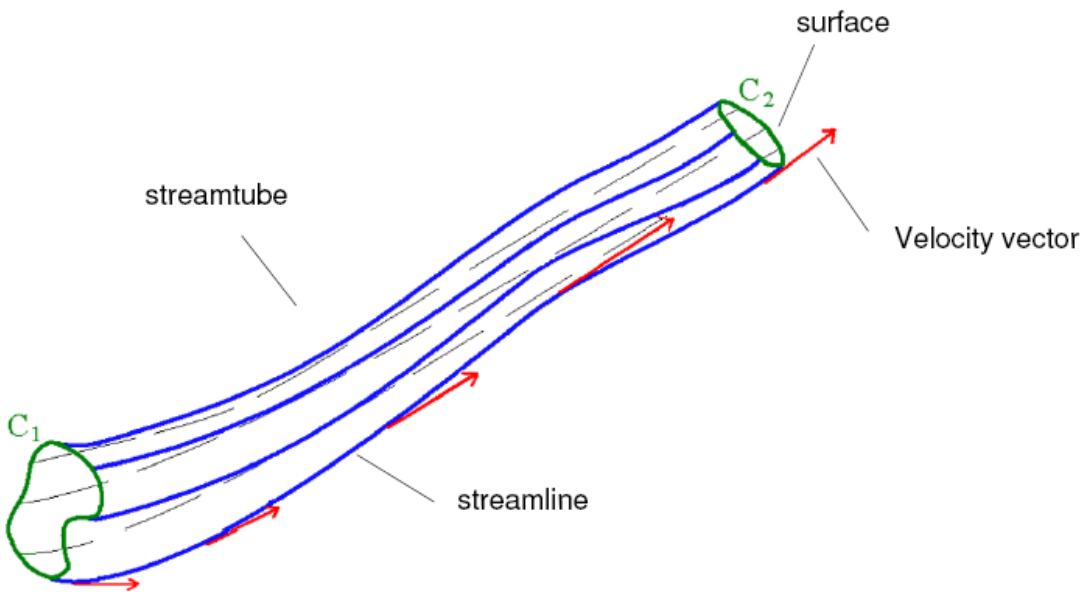
= Eulerian approach (following the movement of different particles at one moment in time)

NOTE 1: two streamlines cannot cross each other!
This would mean that there are two different velocity vectors in one point, which is impossible.
NOTE 2: the boundaries of a flow problem are streamlines



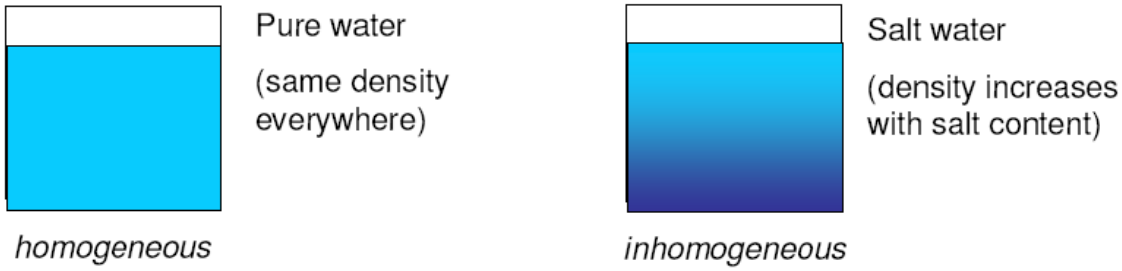


A **streamtube** is a tube formed by streamlines originating and ending on a closed surface. These surfaces form the inlet and the outlet of the stream tube. There is no flow through the walls of a streamtube (see definition streamline).

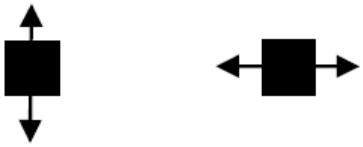


Homogeneity = having the same properties in all points
Inhomogeneity = not having the same properties in all points
In physics, a homogeneous substance is a substance with a solid, consistent with uniform composition and properties. Particles as possible and spread evenly mixed. Each sample of the material will have the same composition and the same features. The opposite of homogeneous is heterogeneous. Here are the various points not all the same features.

Isotropy = having the same properties in all directions
Anisotropy = not having the same properties in all directions
An isotropic material is mentioned as the materialproperties do not depend on the direction. If the propertiesdo depend on the direction, called anisotropic. Further, not all directions the same properties.
Examples:
A homogeneous fluid = a fluid with the same properties in any point.

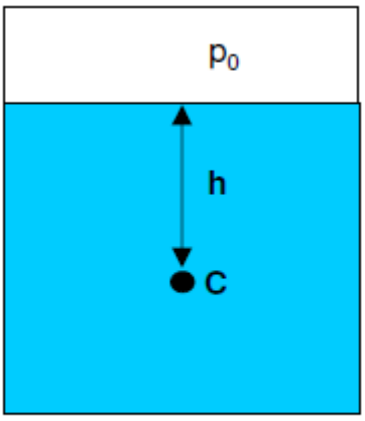


An isotropic fluid = a fluid with the same properties in any direction



7. Give and explain the hydrostatic law, the law of Pascal and Archimedes' law. Also provide the equations and explain the symbols in the equations.

Hydrostatic law



Vessel filled with liquid, in which point C is submerged.

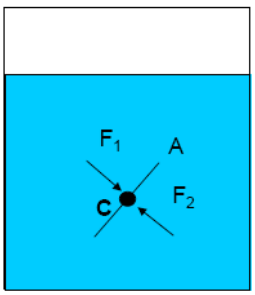
States that the pressure in a point in a fluid is given by the sum of the external pressure and the pressure caused by the fluid column above the point.

$p_c = p_0 + \rho * g * h$ [Pa]

p_0 = external pressure (Pa)
ρ =fluid density (kg/m³)
g =gravitational constant (m/s²)
h =depth of the point below the fluid surface(m)

The law of Pascal
States that the pressure in a point in a static liquid is the same in all directions and only dependent on the density of the fluid and the depth of the point below the fluid surface.

P = rho*g*h [Pa]
rho = fluid density (kg/m³)
g = gravitational constant (m/s²)
h = depth of the point below the fluid surface(m)

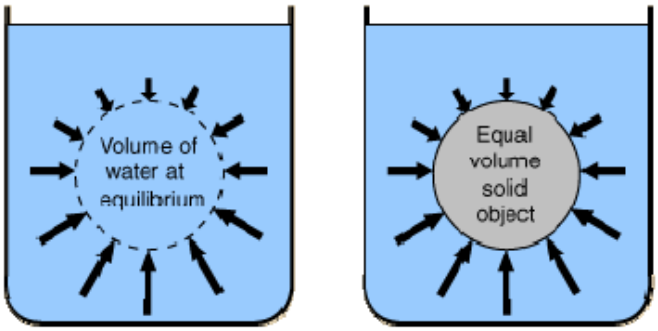


$F_1 = pA = F_2 = pA$

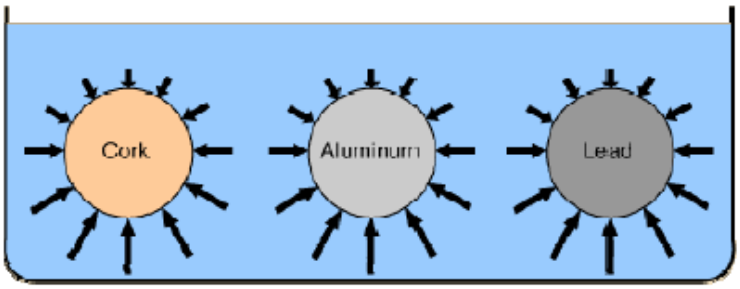
Pressure p can not be indicated with arrows. It is a scalar, not a vector ! Force F is a vector.

F = p*A
F = normal force (N)
P = pressure (Pa)
A = surface area (m^2).

Archimedes' law (wet van Archimedes) or the law of buoyancy states that a body immersed in a fluid is buoyed up by a force equal to the weight of the displaced fluid.
- e.g. balloon in air
- e.g. ship in water



$$F_{arch} = \rho_{water} g V$$



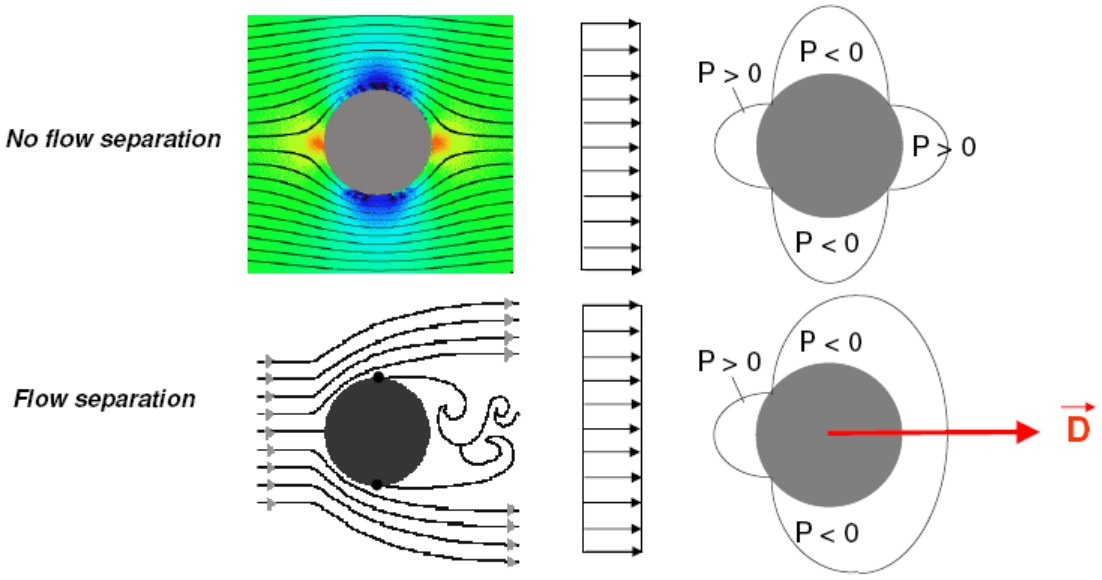
$$F_{arch} = \rho_{water} g V$$

G = $\rho_{material}$*g*V [N]
$\rho_{material}$ = fluid density (kg/m^3)
g = gravitational constant
V = volume of the object (m^3)
($F_{arch}=\rho_{water}*g*V_{underwater}$)

8. Figure 1 shows turbulent flow around a vertical cylinder in a horizontal cross-section.
The flow direction is indicated with the arrow. What type of lines are the black lines? What is the meaning of the reduced or increased spacing between these lines? Is this figure realistic? If not, what is incorrect, and what are the consequences for the flow pattern and the pressures and forces?

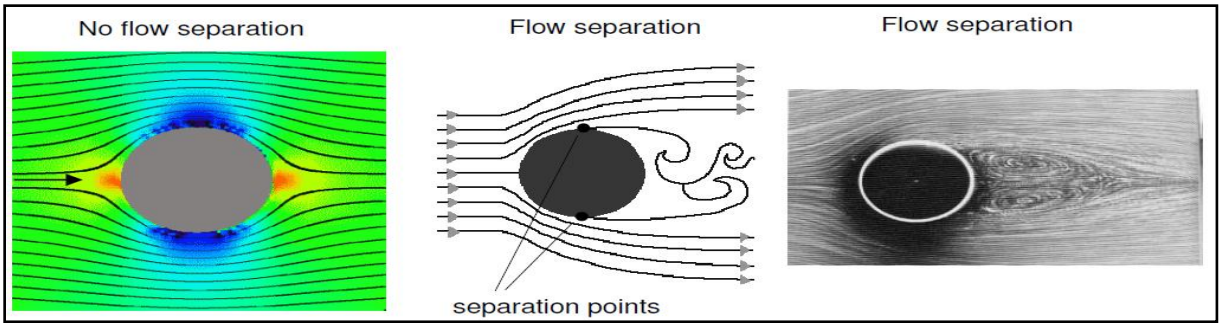
Black lines type
The black lines indicate the streamlines of the liquid again. The spaces between the streamlines indicate the speeds of the current. If the lines are close together (high density of lines) can find a local high speed. If the lines further apart from each other, there is a lower speed. Bernoulli's law states that pressure is inversely proportional to the speed. This creates low pressure at high speeds and vice versa. What we see in the picture is that the rates increase sharply at the top and bottom of the cylinder. This creates a reduced pressure on these points (Bernoulli). Where the power comes from the cylinder speed is low and therefore very much pressure, this also happens at the back. The picture shows a cylinder

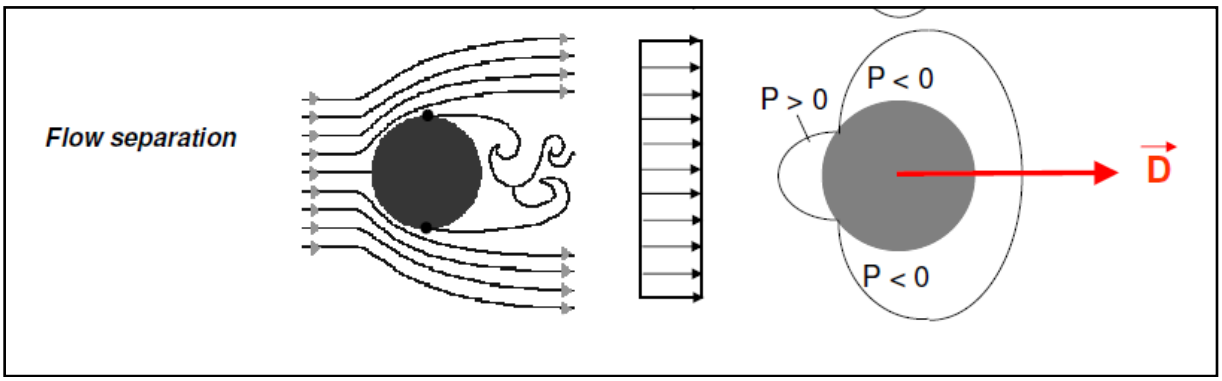
shown in a perfect flow (μ = 0). The electricity that comes out has a constant speed and turbulence.



Realistic figure?

In reality the current is not flowing as presented in the picture. In the figure assumes a perfect stream, mu = 0, constant velocity and turbulence. In reality, the flow separated at a point on the cylinder. This is because the velocity of air at the surface is 0. The air is laminar and be much faster regardless of the surface than turbulent flow. This creates a vortex.





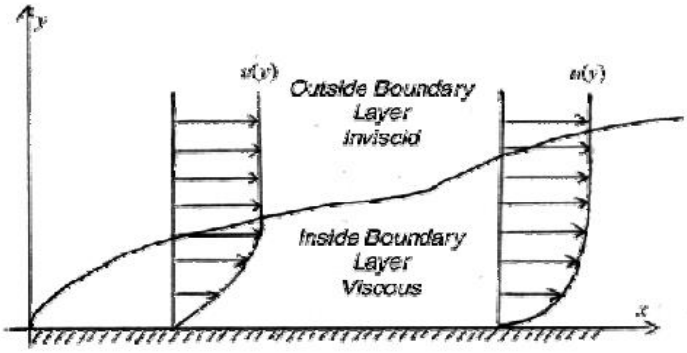
In this wake, the high pressure region is not created by this flow Separation. Instead, it creates a low pressure area at the rear of the cylinder, which is a low power current (downstream Force) develops.

The difference between the prediction of Euler and the reality is caused by the existence of the viscosity and the fact that the fluid does not slip over the surface. Instead, the speed of the liquid over a surface 0. The layer of liquid in the vicinity of the binding surface, in which

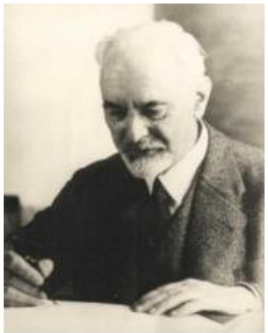
the velocity of the fluid quickly rises from 0 at the surface to a constant value, the boundary layer (boundary layer) mentioned.

The difference between the prediction by the Euler equations and reality is caused by the existence of viscosity μ and the fact that the fluid does not slip over the surface. Instead, the fluid speed at a surface is zero (non slip condition).

The layer of the fluid in the immediate vicinity of a bounding surface, in which the fluid speed rapidly increases from zero at the surface to a constant value, is called the boundary layer.



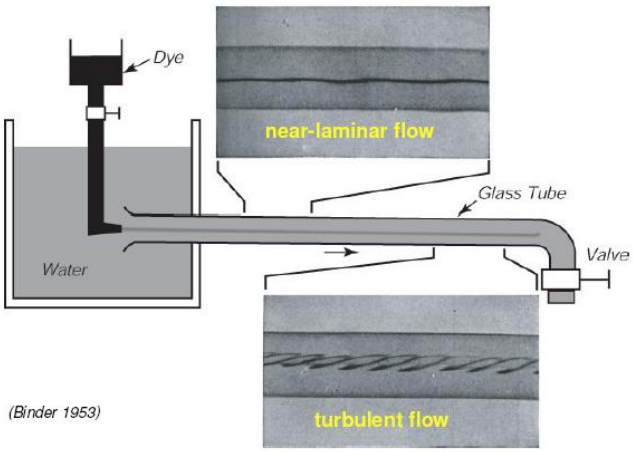
The boundary layer concept is due to Ludwig Prandtl (1875 – 1953).



With a smooth ball that flies through the air, the air flows in layers parallel to the surface of the ball. Something like this is called laminar flow. Such air flow is not as good for the flight speed of the ball. The passing air 'sticks' as it were on the ball and that causes an extra resistance. The hole pattern on golf balls disrupts this stratified flow so that air close to the spinal little small ball forms. This is called turbulent flow. Turbulent boundary layers tend to have less extra resistance.

9. Explain the experiment done by Osbourne Reynolds. Give the definition of the Reynolds number. What is the meaning of this number? What are its advantages and disadvantages?

Introducing dye into the flow through a glass tube of constant section. For low velocities, the dye flowed through the tube without significant mixing with the surrounding water flow. At higher velocities and/or further downstream, mixing occurred.



Reynolds number represents the ratio of inertia forces to viscous forces:

$$Re = \frac{VL}{v}$$

Where V is a characteristic velocity scale, L is a characteristic length scale and n is the kinematic viscosity of the fluid.

A high Reynolds number means that the inertia forces are more important than the viscous forces. Such situation can be representative of turbulent flow.
A low Reynolds number means that the viscous forces are more important, which can be representative for laminar flow.
The transition from laminar to turbulent flow does not occur at a fixed Re number. In between the laminar and turbulent regimes, there is a transitional regime, in which the flow is neither laminar, nor turbulent.

Inertia is the resistance of any physical object to a change in its state of motion or rest
Viscosity is a measure of the resistance of a fluid which is being deformed by either shear stress or tensile stress.

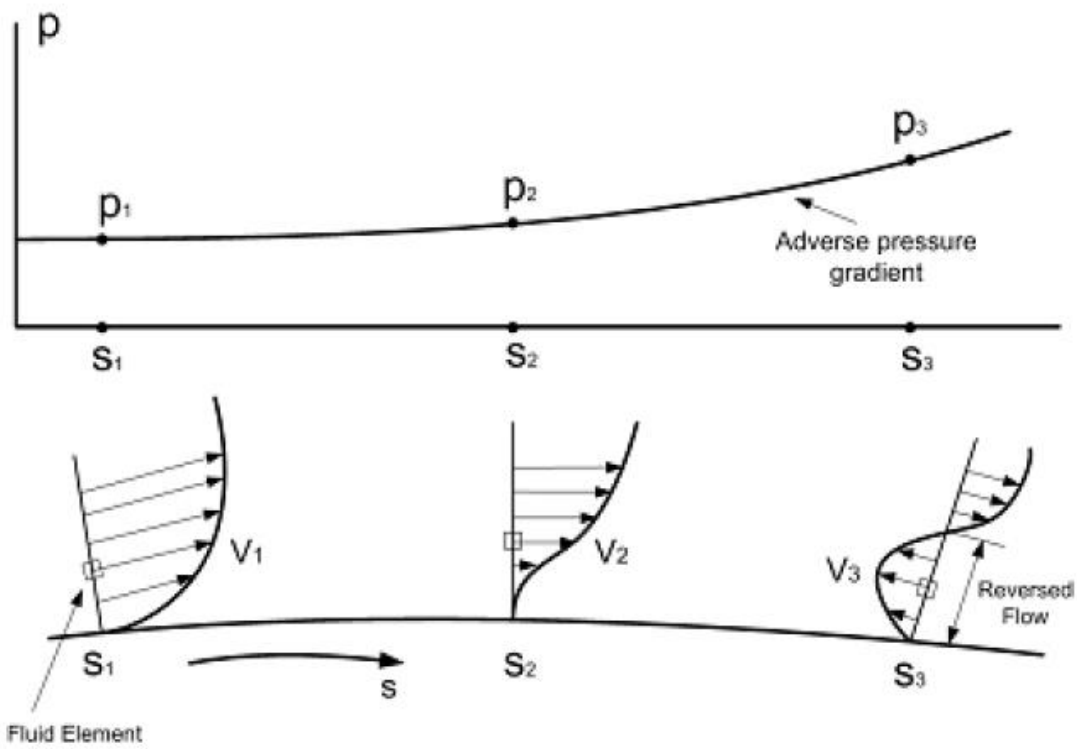
Benefits
An advantage of the Reynolds number that can be determined whether a flow is laminar or turbulent. This allows the properties of the flow can be estimated.

Disadvantages
- It is often not clear what should be taken as the characteristic length scale of the current. For example, for wind flow over a building there are a number of options: the height of the building, the width of the building, the height of the ABL approach the building.
- Another disadvantage is that the change from laminar to turbulent flow does not always take place at the same Reynolds number or Reynolds number fixed it. There is a transition zone where the flow is laminar and not turbulent. Additionally, the Reynolds number of this transition zone may be quite different for each type of flow.

10. What is a boundary layer? What is flow separation? What is the reason for flow separation? Using a drawing, explain at which positions of the surfaces of a building flow separation occurs.

Boundary layer

The layer of the fluid in the immediate vicinity of a bounding surface, in which the fluid speed rapidly increases from zero at the surface to a constant value. Flow separation leads to the formation of a wake, a region of low pressure extend. Flow separation occurs due to an adverse pressure gradient. Turbulent boundary layers show better attachment to the surface. The further the separation points are located downstream (rougher surface), the smaller the area of the ball on which the under pressure in the wake is exerted.

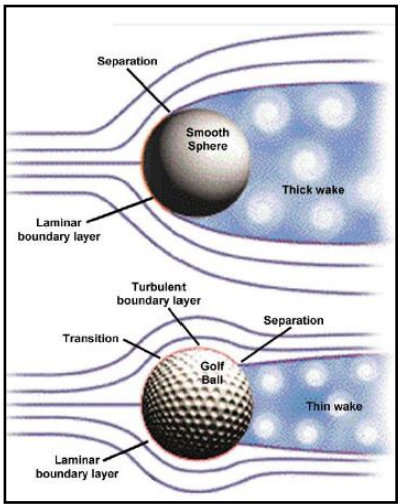


Flow separation occurs around the corners of a building (8). It also depends on the roughness of the surface.

Flow separation occurs when the boundary layer far enough from an adverse pressure gradient (an adverse pressure gradient occure When The static pressure increases in the direction of the flow) down the speed of a boundary layer drops to 0. The flow is detached from the surface of the object, and instead takes the form of eddies (an eddy is the swirling of a fluid and the reverse current created When The fluid flows fits an Obstacle) and vortices (A vortex (plural : vortices) is a spinning, or at turbulent flow of fluid). In aerodynamics, flow separation may result in an increased drag (drag (Sometimes called air resistance or fluid resistance) refers to Forces That Oppose the relative motion of an object through a fluid (a liquid or gas). Drag forces act in a direction opposite to oncom the flow velocity), pressure drag which is caused mainly by the pressure difference between the front and back of an object, where the current flows.

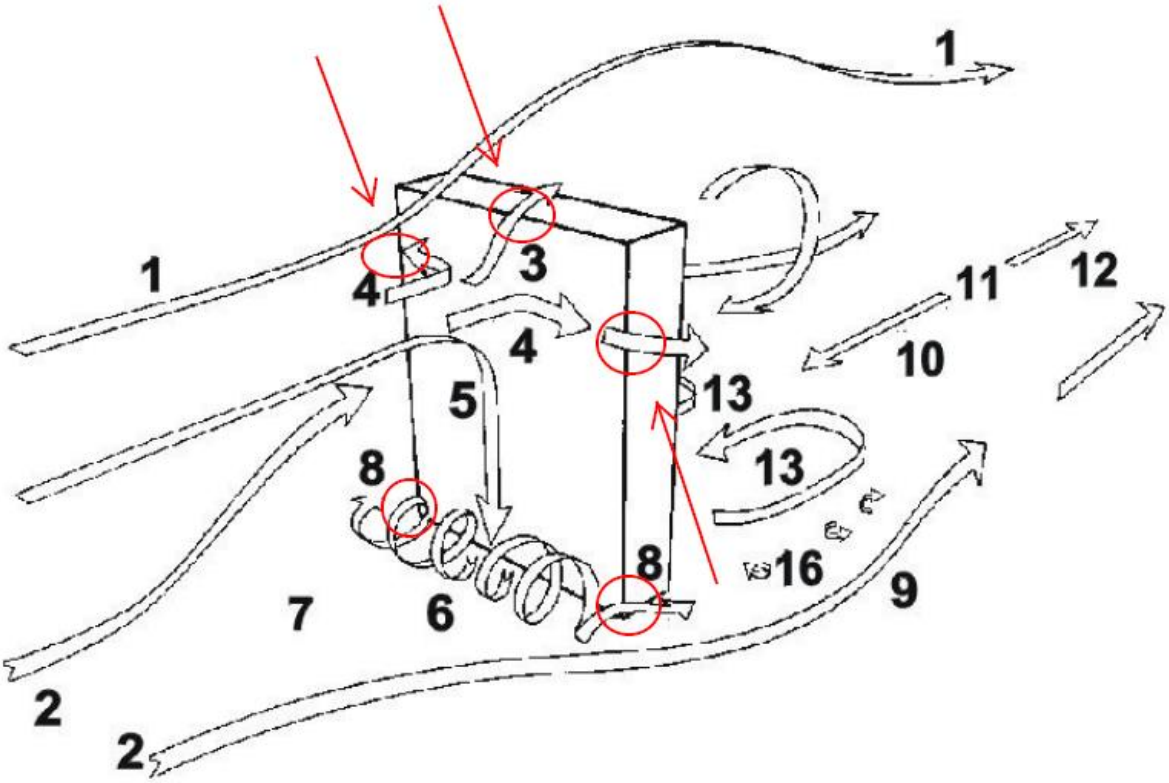
This flow separation should be prevented as long as possible, because the power, as long as possible the object is retained. This has the advantage that the object can flow faster. In laminar flow the flow separation takes place. When the turbulent flow around the object so created and held, there is less flow separation occurs. This happened with golf balls, the surface contains holes. This creates a turbulent flow around the object. The further the separation points are positioned, the smaller the area where the ball under pressure in the wake of being tense.

In the case of a ball with a smooth surface, the airflow in the thin layer right next to the ball (called the boundary layer) is very smooth. This type of flow is called laminar. For a ball with a smooth surface, the boundary layer separates from the ball's surface quite early, creating a wide, turbulent wake pattern behind the ball. The turbulent wake exerts a drag force on the ball. When dimples are added to the surface of the ball, they create turbulence within the boundary layer itself. The turbulent boundary layer has more energy than the laminar boundary



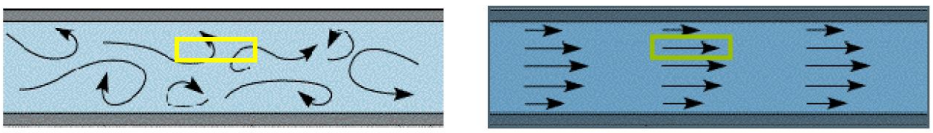
layer, so it separates from the surface of the ball much later than the laminar boundary layer flowing over the smooth ball (Figure 2, bottom). Since flow separation occurs later, the turbulent wake behind the ball is narrower, resulting in less drag force.

Flow separation occurs at the **corners of a building**.



11. Give three different definitions of turbulent flow, and explain which one of these is the best and why.

Turbulent flow is a fluid regime characterized by chaotic, stochastic property changes. This includes low momentum diffusion, high momentum convection, and rapid variation of pressure and velocity in space and time.



The next 3 definitions are given:

- “Turbulence was the invention of the Devil on the seventh day of creation, when the Good Lord wasn’t looking” (Bradshaw 1994)

- “A type of fluid (gas or liquid) flow in which the fluid undergoes irregular fluctuations, or mixing, in contrast to laminar flow, in which the fluid moves in smooth paths or layers. In turbulent flow the speed of the fluid at a point is continuously undergoing changes in both magnitude and direction.” (Encyclopedia Brittannica)

- Turbulent flow = Chaotic flow where fluid particles are exchanged between different fluid layers

Which is the best?
There is not really one good accepted definition of turbulent flow. All these definitions do not really give away what it actually means. It can be defined best by summarizing its properties. This is done in the following numeration (Ferziger & Peric 1997):
1. Turbulent flows are highly unsteady (they “appear” random)
2. They are 3-dimensional (even if mean flow is 2D)
3. Large amount of vorticity present
4. Conserved quantities are stirred, mixed: turbulent diffusion (exchanging parcels of fluid)
5. Mixing is a dissipative process (kinetic energy to internal energy)
6. Coherent structures are present
7. Fluctuations occur on a broad range of length and time scales

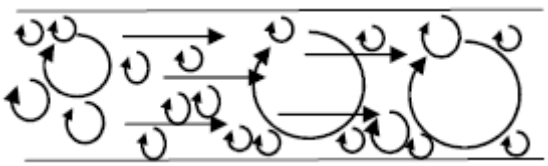
This can be compared best with the second definition. The first definition is from a frustrated investigator, who could not find a good definition for turbulent flow. The third definition is from a dictionary and is very insufficient in content. In the second definition most properties are displayed and it is therefore the best of the 3.

12. Explain the three main categories for predicting turbulent flow and the differences between these categories.

Several methods exist for predicting turbulent flows with CFD. Three most popular approaches in engineering are:

DNS: direct numerical simulation
Is the approach in which the Navier-Stokes equations are solved for all of the motions in the turbulent flow. Since all scales of the turbulence are solved, this method is “exact” but it puts very high demands on computational resources. Currently, DNS is possible only at low Reynolds numbers and for relatively simple geometries. It is used for studying and understanding turbulence and for the verification of RANS and LES turbulence models.

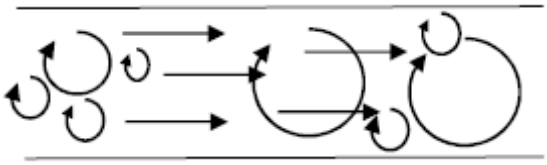
- Solve the exact Navier-Stokes equations completely
- All vortices/eddies are “solved”, nothing “modelled”
- Exact
- Very time-consuming, huge computational resources, only very simple geometries, huge amounts of data



LES: large eddy simulation
In this method, the Navier-Stokes equations are filtered. This means that only the small turbulent eddies (that are smaller than the size of a filter that is usually taken as the mesh size) are removed. The largest-scale motions of the flow are solved, while the small-scale motions are modelled: the filtering process generates additional unknowns that must be modelled in order to obtain closure. Also here, turbulence models are used. Because they are only used to model the scales of turbulence smaller than the size of the filter (grid), they are called “subgrid-scale models”. Because this method solves more of the turbulence and models less (compared to RANS), the accuracy of the simulation is improved, however at the

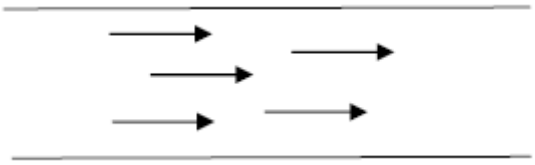
expense of increasing computer requirements. Important research work in this field is ongoing and in an increasing number of cases, simulations of wind flow around buildings with LES are conducted.

- Solve the “filtered” Navier-Stokes equations
- Only the large eddies are “solved”, the small ones are “modelled”
- Not exact, but less computationally demanding



RANS: Reynolds averaged Navier-Stokes
These equations are derived by averaging the Navier-Stokes equations (time-averaging if the flow is statistically steady or ensemble-averaging for time-dependent flows). With the RANS equations, only the mean flow is solved while all scales of turbulence have to be modelled (i.e. approximated). The averaging process generates additional unknowns (the Reynolds stresses) and as a result the RANS equations do not form a closed set. Therefore approximations have to be made. These approximations are called turbulence models (e.g. k-e models). The RANS method is the one that has been most widely applied and validated in the field of numerical computation of wind flow around buildings and air flow inside buildings.

- Solve the “averaged” Navier-Stokes equations
- Only the mean flow is “solved”, all eddies are “modelled”
- Not exact, less accurate, but generally applicable
- In the RANS approach, the “effect” of turbulence on the mean flow is modelled



approach	solve	model (appr.)
DNS	All eddies	Nothing
LES	Large eddies	Small eddies
RANS	Average flow	All

exact

Turbulence model needed

13. Explain the two typical problems associated with the use of the k-ε model for predicting turbulent flow around buildings.

main defects of the k-ε model to calculate the anisotropic high wind flows around buildings. These defects are mainly caused by the inaccurate prediction of the turbulent kinetic energy levels:

1. The standard k-ε model overestimates the production of kinetic energy k around the corners at the front and sides. This exaggerated levels of turbuelente diffusions arrested. A typical result is that the standard Ke model of separation and circulation areas underestimated arising on the roof of the side where the wind is. In the worst case, these regions not predicted.
2. The standard k-ε model underestimates the value of K in the recirculation region behind the building and thus the speed and size of the recirculation vortex overestimated.

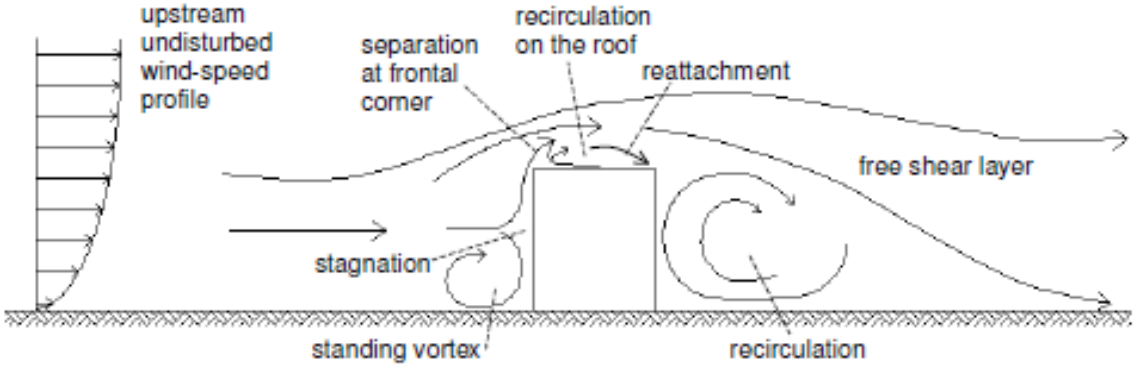


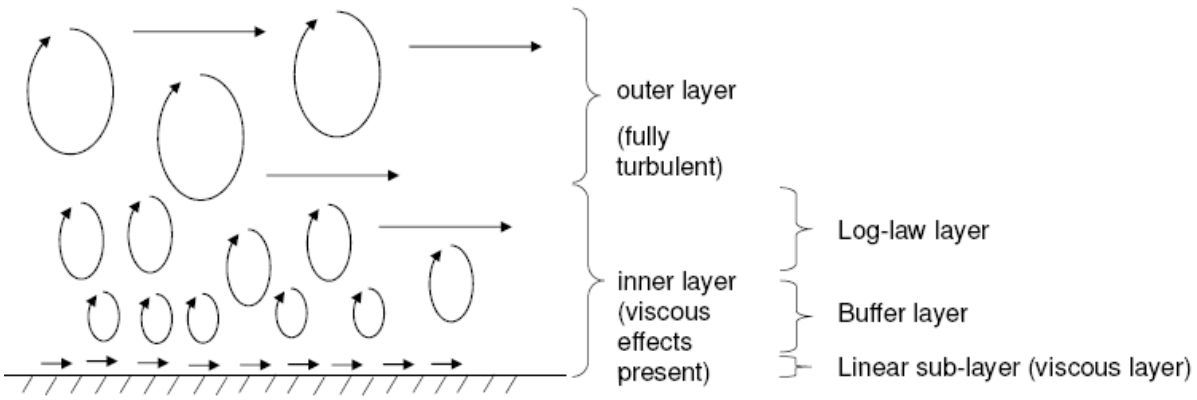
Figure 3.5. Schematic illustration of the wind-flow pattern over a building including the regions where the k-ε model shows deficiencies: the separation at the frontal corner, the recirculation on the roof and the recirculation (wake) behind the building.

14. Explain the law of the wall and give a figure. Why is this law of the wall important in CFD?

Importance for CFD
The k-ε model is useful only for turbulent core flows, for example in areas where the flow is fully turbulent. In regions close to solid walls, the Reynolds number are small and the viscous effects important. The flow close to walls therefore need special attention. Accurate modeling of the area near the wall is important for accurate predictions of the flow field.

The law of the wall states that the average velocity of a turbulent flow at a certain point is proportional to the logarithm of the distance from that point to the "wall", or the boundary of the fluid region. It is only technically applicable to parts of the flow that are close to the wall (<20% of the height of the flow), though it is a good approximation for the entire velocity profile of natural streams.

Different flow behavior in different layers => different treatment necessary in each layer.
We need to know the extent (height) of each layer!



Log-law layer: inertial effects are dominant over viscous effects
Buffer layer: intermediate layer between the linear sub-layer and the log-law layer where the viscous and turbulent effects are about equally important
Linear sub-layer (viscous layer): very close to the wall: viscous effects dominate the flow
The outer layer: free from viscous effects. Another law is valid: the velocity defect-law. The law of the wall is important because closer to the wall the viscous forces become dominant. Closer to the wall in turbulent flows different regions can be distinguished with each a different treatment. Therefore we need to know the extent of each layer (and is this law important).

In regions near solid walls, the local Reynolds number will be small and viscous effects are important. So a local, wall-based Re-number is needed.

U= characteristic velocity
y = distance from the wall
V = kinematic viscosity
the local wall-based Reynolds number Rey. The definitions are as follows:

$$Re_y = \frac{U\,y}{\nu}$$

For wind flow around buildings, the characteristic length scale can e.g. be taken as the height of the building. This indicates that at this scale (far from the wall, in the turbulent core of the flow), the inertia forces (UL) are much larger than the viscous forces (n). On the other hand, when we approach the wall, y and thus the wall-based Reynolds number Rey decrease. Close to the wall, where Rey is of the order of one, the inertia forces and the viscous forces are about equal in order of magnitude. Closer to the wall, the viscous forces become dominant. This indicates that in turbulent flows along walls, different regions can be distinguished. A distinction is made between the linear sub-layer, the buffer layer, the log-law layer and the outer layer.

$Re_y >> 1$ inertia forces are much larger than viscous forces (turbulence dominates)
$Re_y = 1$ the inertia forces and the viscous forces are about equal in order of magnitude.
$Re_y < 1$ viscous forces become dominant.
Different regions can be distinguished.

The logarithmic law of the wall is a self similar solution for the mean velocity parallel to the wall, and is valid for flows at high Reynolds numbers - in an overlap region with approximately constant shear stress and far enough from the wall for (direct) viscous effects to be negligible:

The thickness of the linear sub-layer of a boundary layer decreases as the Reynolds number increases. At high Reynolds numbers in complex, three-dimensional flows (Re » 105), wall functions are often the only possibility. In such cases, it becomes almost impossible to resolve the very thin linear sub-layer.

15. Explain wall functions and low-Reynolds number modelling. What are the main advantages and disadvantages associated with each of these techniques?

Wall Function Method

Wall functions are a set of semi-empirical formulas and functions that are used to the deviations of the k-model ε discover. With these formulas and functions may appear relatively good flow estimates are made near a wall. The wall functions to link the values at the center point of the wall adjacent to the cell wall and thus provide boundary conditions for the RANS equations and the transport equation of K

In this method, the inner layer (inner layer) is not resolved. Semi-empirical (experience-based) functions are used to the wall and the total turbulent region and to link the turbulent models valid for turbulent core flows to be used with this wall features.
This method is point P, for example, the center point of the wall adjacent cell or control volume, the first cell from the wall where the flow is resolved, should be in the log layer. This method can work with relatively large cells, course mesh near the wall.

Low Reynolds number modeling method

In this method, the turbulence model adapted to the modeling of the inner layer (inner layer) all the way down as possible. In this method, the point P in the linear sub-layer lying. This method requires a very fine grid near the wall.

The figure below shows how these methods are going to force a volume controlled method.

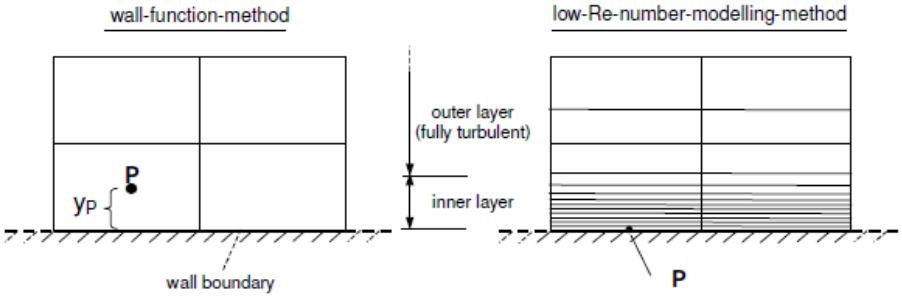


Figure 3.8. Spatial discretisation in the control-volume method for near-wall modelling using the wall-function method and the low-Re-number-modelling method. The former allows a relatively coarse mesh near the wall (point P of wall-adjacent cell must be situated in the log-layer), while the latter requires a dense mesh near the wall (point P must be situated inside the linear sub-layer).

The thickness of the sub-layer decreases when the Reynolds number increases. At a high Reynolds number in complex 3D flows, it is often impossible to work with the low Reynolds number method. Therefore, in such cases work with the wall function method. Wall functions have some important advantages over low Reynolds number modeling:
- Economic cheaper (less CVS, less computer time and storage)
- They are also the introduction of additional empirical information in specific situations, such as highly textured walls.

Disadvantages
- The wall function will fail if the flow conditions differ too much from the ideal conditions that are assumed in the equations. This may be the case for flows that are separated in the separation and re-attache the recirculation zones and regions. The fact that the wall function can not be applied here is often ignored and it is therefore still used. This can produce large errors occur. There are other wall features developed.
- The wall function is less accurate than the low Reynolds number method. The wall function is often accurate enough, so it is often used (wall functions give reasonably accurate estimates for the majority of high Reynolds numbers).

The near-wall physics have to be taken into account in modelling the near-wall region.
Two main options exist:
(1) Wall functions
(2) low-Reynolds number modeling

For both approaches, special guidelines must be followed to obtain accurate results

<u>Mesh requirements</u>

Wall functions: The distance from the wall to the centre of the wall-adjacent cell must be matched with the region where the log-law is valid: <u>y_P must be situated in log-law layer.</u>

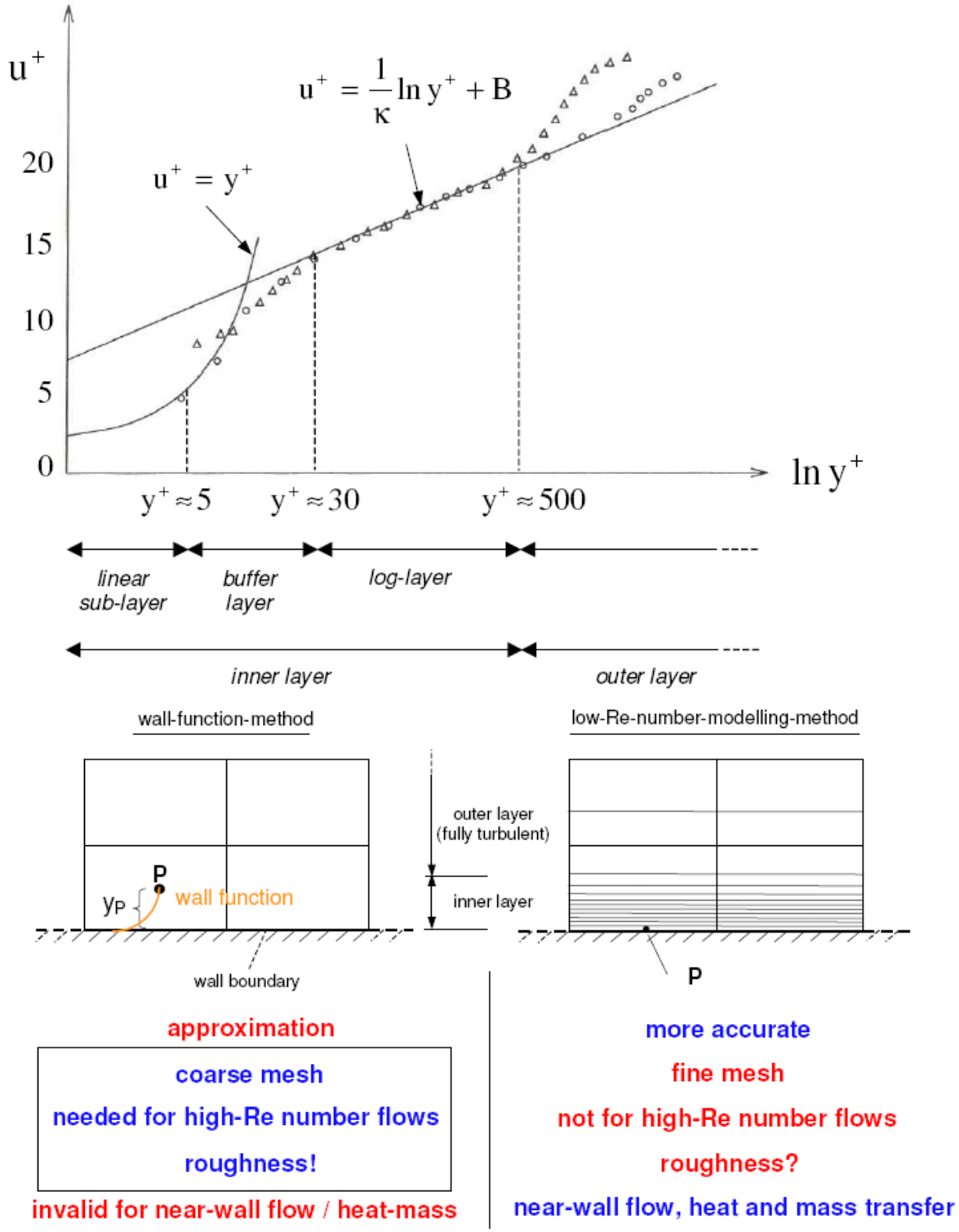
$$y^+ = \frac{\rho u_\tau y_p}{\mu}$$

$30 < y^+ < 60$; preferably $y^+ \approx 30$ (Fluent)
Preferably $30 < y^+ < 1000$; y^+ should not exceed 1000 (Ansys)
$y^+ \approx 50$ (NASA)

Low-Re number modelling: Fine cells are needed to resolve the near-wall region. <u>y_P must be situated in linear sublayer.</u>

$$y^+ = \frac{\rho u_\tau y_p}{\mu}$$

$y^+ < 4$ to 5; preferably $y^+ \approx 1$ (Fluent)



The thickness of the linear sub-layer of a boundary layer decreases as the Reynolds number increases. At high Reynolds numbers in complex, three-dimensional flows (Re » 10^5), wall functions are often the only possibility. In such cases, it becomes almost impossible to resolve the very thin linear sub-layer.

16. Give and explain the five errors in CFD and give examples. How can they be determined and reduced?

1. Physical modeling defects (physical modeling errors)
Physical modeling defects are defects caused by uncertainties in the formulation of a model and by applying simplifications of the model. Based on the preceding paragraphs of this text may be a number of examples of physical modeling defects:

- The RANS equations run with a given turbulence model
- The EVM (Eddy viscosity model)
- The use of specific constant values in the Ke model
- The use of wall functions
- Modelling the roughness of the surface Fluent
- simplifications of the geometry of the model

Physical model defects can be assessed by validation studies be undertaken that focus on certain models and some applications (such as the implementation of the Ke model in predicting the wind flow around buildings). Validation means comparing simulations with high accuracy measurements to determine the validation.

2. Computer round-off errors
Round-off errors are not considered significant when compared with other errors. If they are suspected to be significant, one can perform a test by running the code at a higher precision.

3. Iteration-convergence error (user related)
This error is introduced because the iterative procedure has to be stopped at a certain moment in time. Iteration-convergence errors can be estimated, see text. Guidelines are provided further.

4. Discretisation error (user related)
Discretisation errors are generated from representing the governing equations and the equations of the turbulence model on a mesh that represents a discretised computational domain (for unsteady calculations also time discretisation causes discretisation errors). This type of error is also called “numerical error”. <u>Grid sensitivity analysis is a minimum requirement in a CFD simulation</u>.

5. Computer-programming errors
Program failures are errors made in writing the computer code. These types of defects can be discovered by systematically carrying out verification and validation studies and in studies comparing the results of a particular code with the results of a similar code.

It is noted that validation and verification are terms that have a specific meaning in the context of CFD simulations. This means that the meaning of these terms may differ from those normally used in other areas of science. The definition given in the AIAA 1998 is: validation refers to the identification and quantification of defects by comparing simulation results with data from experiments.

Authentication refers to the identification and quantification of defects in the model and the solution. There are two aspects to the versificeren: verification of the encoding and verification of the calculation. Verification of the code also has the removal of computer programming 'securities. Verification of the calculation is to determine the iteration convergence 'securities and discretisation effects of a CFD simulation. Comparison of the CFD simulation results with high precision results, for example with analytical solutions or DNS data, is also a way of verification.

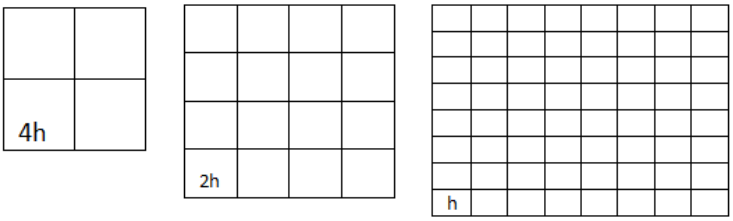
It is noted that more validation is the responsibility of an engineer than a mathematician, while the opposite is true for authentication. Validation is used for modeling physical defects. Authentication is used for the other four types of defect.

17. Give the equation for the discretisation error ε_h^d d on a grid with spacing h. Explain the variables in this equation. What is Richardson extrapolation?

$$\varepsilon_h^d \approx \frac{\varphi_h - \varphi_{2h}}{2^P - 1}$$

φ_h= exact solution of discretised equations on a mesh with reference spacing h (-)
p= order of discretisation scheme (-)
h= reference grid spacing (m)

Goal
To find a grid-independent solution (to find a mesh where the result stays the same).
Het vinden van een grid-onafhankelijke oplossing (het vinden van een mesh waarbij het resultaat hetzelfde blijft).



$\Phi=\varphi_{4h}+\varepsilon_{4h}^d$ $\Phi=\varphi_{2h}+\varepsilon_{2h}^d$ $\Phi=\varphi_h+\varepsilon_h^d$ → 3 equations, 3 unknowns.

Richardson extrapolation
A Richardson extrapolation can be applied to obtain an approximation of the exact solution Φ that's more accurate than the solution φ_h on the finest grid. This method simply consists of adding the estimate of the discretisation error to φ_h.
*In 3D it's not easy to refine the grid with a factor 2, often chosen for √2. Also a factor 2 will make the grid too fine to be handled. In reality often comparing of three grid refinements, no estimating of the discretisation error.

18. You are the head of a building design office. One of your employees has made a CFD simulation to assess pedestrian wind nuisance around a new building that you have designed. What are the criteria for you in order to believe that his or her results are accurate and reliable? Explain why.

Results can only be trusted / used if they have:
1. Been performed on a mesh obtained by grid-sensitivity analysis
2. Been performed taking into account the proper guidelines that have been published in literature
3. Been carefully validated
    - Wind flow over a roughness change
    - Wind measurements and instruments

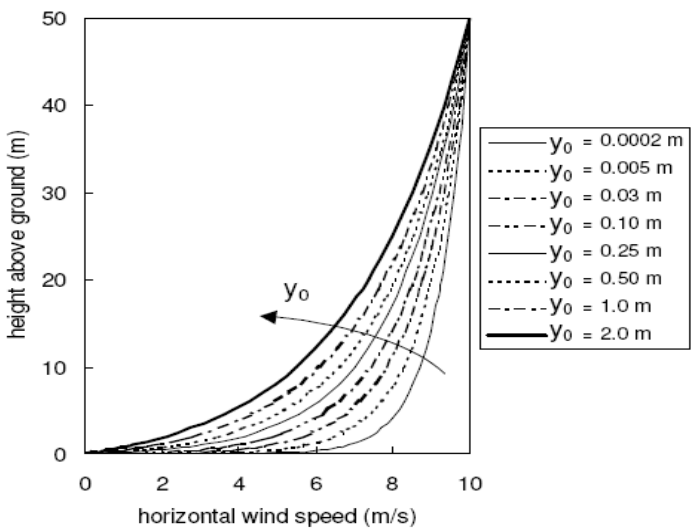
With validation one shows the assembly between reality and simulation. If the proper guidelines are known then a simulation can be performed again by others if they want to. Also every model, like the k-ε model, has their own guidelines and not every model is usable in every simulation, so it is necessary to follow the guidelines related to the model. A grid-sensitivity analysis is also a must, and done by refining or coarsening the mesh (grid) by a

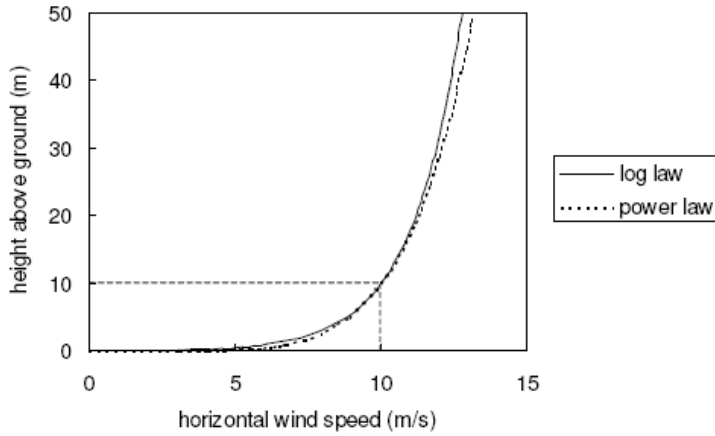
constant factor to obtain a grid-independent solution. Computer programs can check the grid too, for a first insight in the grid.

19. What is the atmospheric boundary layer? Draw the shape of the wind speed profile, turbulence intensity profile, turbulent kinetic energy profile and turbulence dissipation rate profile in the atmospheric boundary layer as a function of height. Explain why these profiles have this particular shape.

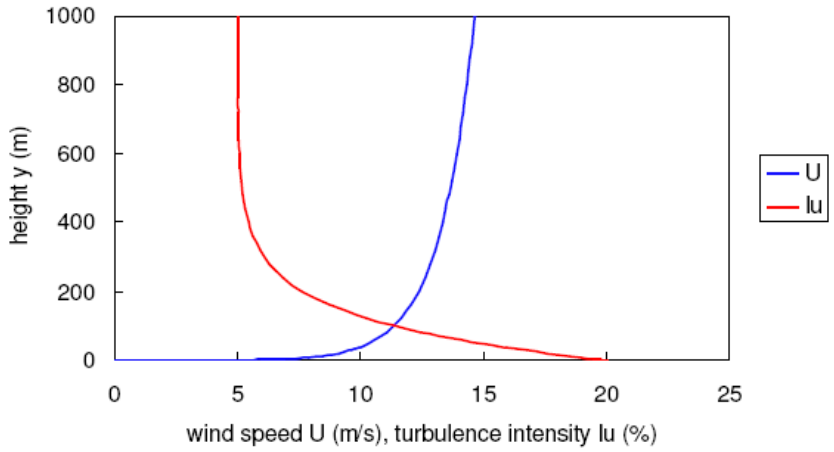
The **atmospheric boundary layer (ABL),** is the lowest part of the atmosphere and its behavior is directly influenced by its contact with a planetary surface. In this layer physical quantities such as flow velocity, temperature, moisture etc., display rapid fluctuations (turbulence) and vertical mixing is strong. Above the ABL is the "free atmosphere" where the wind is approximately geostrophic (parallel to the isobars) while within the ABL the wind is affected by **surface drag** and turns across the isobars. The free atmosphere is usually nonturbulent, or only intermittently turbulent.

Wind speed profiles



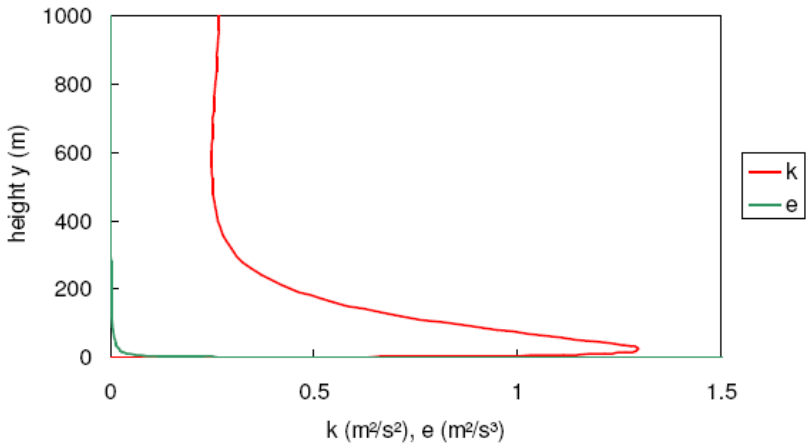


Turbulence intensity profile



The higher you are, the fewer buildings or other obstacles are present. These obstacles make for a true turbulent flow. At ground level there is thus much more turbulence than are present at higher levels.

Turbulent kinetic energy profile (rood) en Turbulence dissipation rate profile



A lot of turbulence means a lot of kinetic energy. When there's no dissipation of energy the shape of the turbulent kinetic energy follows the shape of the turbulence intensity. When there's dissipation at one height (related to the roughness length) the turbulent kinetic energy is dissipated so the turbulent kinetic energy profile decreases to zero. At 0m there's no flow and no turbulence, so no kinetic energy. The shape of the decrease of the turbulent kinetic energy curve is related to the shape of the dissipation curve.

20. Give the equations of the logarithmic law and the power law for mean wind speed in the atmospheric boundary layer. Explain the variables in these equations. Explain in detail how the aerodynamic roughness length can be determined.

<u>Logarithmic law</u>

$$U(y) = \frac{u^*_{ABL}}{\kappa} \ln\left(\frac{y + y_0}{y_0}\right)$$

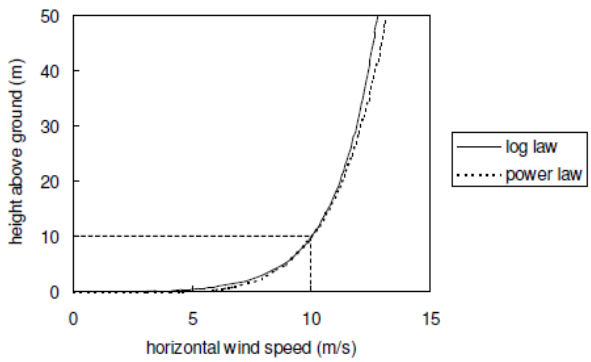
where

$U(y)$ is wind speed at height y
u^*_{ABL} is friction "velocity"
κ is the Von Karman constant (= 0.42)
y_0 is the aerodynamic roughness length

Power law

$$\frac{U(y)}{U_{ref}} = \left(\frac{y}{y_{ref}}\right)^{\alpha}$$

where
U(y) is wind speed at height y
U_{ref} is the reference wind speed at height y_{ref}
α is the power-law exponent

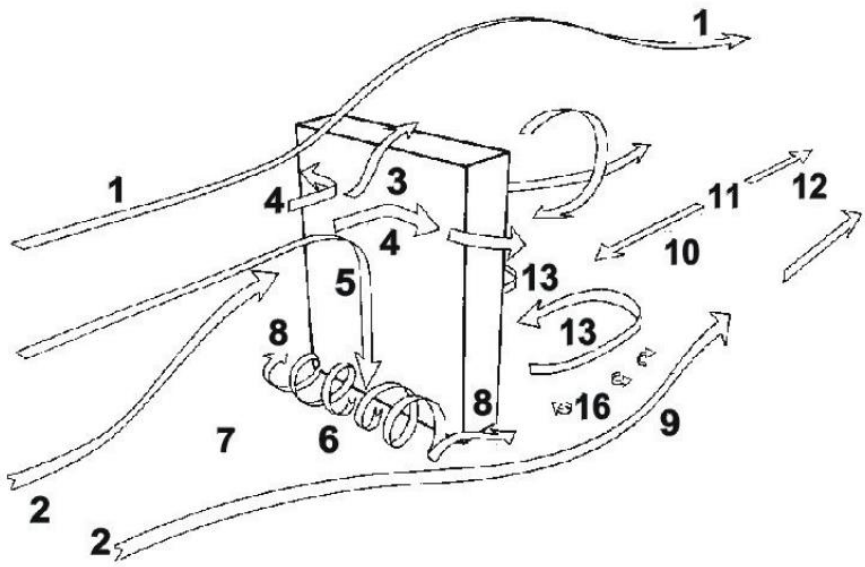


A direct relation exists between y_0 and α, e.g.:

y_0 (m)	α
0.03	0.17
1	0.28

Aerodynamic roughnesslenght
y0 is dependent on the nature of the crude to the plane elements; size, shape, orientation, and distance It's not really a height, is an "equivalent roughness that is felt by the flow".It is a measure for the size of swirls on the surface. It can meet the of Davenports roughness classifictation be established. This gives each y0 Description of the scenery and makes it possible, on the basis of a visual image to the to determine the aerodynamic roughness length

21. Make a drawing of the three-dimensional wind-flow pattern around a high-rise rectangular building and indicate and explain the main flow features. Which flow features are important for pedestrian wind conditions?



1. Flow over building
   - Wind will flow against the building, where part of this flow moves over the building.
2. Oncoming flow
   - Another part of the fore mentioned flow will go around the building its edges.
3. Flow from stagnation point over building

- On the biggest surface façade perpendicular to the wind direction, there is a stagnation point at about a height of 2/3th of the façade. There exists maximal pressure at this point. Because of this, the flow will split up. 1/3th goes up (3) and 2/3th goes down (5).
- On top of the building, flow will transcend the corners, flow separation takes place.

4. Flow from stagnation point around vertical building edges
5. Down flow from stagnation point
6. **Standing vortex, base vortex or horseshoe vortex**
   - The downward flow produces a vortex on ground level.
7. Stagnation flow in front of building near ground level
   - A stagnation flow is caused by the original wind flow which collides with the standing vortex.
8. **Corner streams (vortex wrapping around corners)**
   - The standing vortex wraps around the corner and creates high wind speeds
9. Flow around building sides at ground level (adding to corner streams)
10. Recirculation flow
    - On the backside of the building an under pressure zone is created, this results in an articulating flow
11. Stagnation region behind building at ground level.
12. Restored flow direction
13. Large vortices behind building
    - The recirculation flow (10) is responsible for creating slow rotating vortices behind the building.
16. Small vortices in shear layer
    - Between the vortices (13) and the corner streams (8,9) there is a zone with a high speed gradient which creates small vortices

The standing vortex (6) and the corner streams (8) are important pedestrian wind conditions which are influenced by the building dimension, wind speed and wind direction. Some important general guidelines can be extracted:

1. Building entrances near corners of especially high-rise buildings should be avoided, as well as walkways or bicycle routes. In addition to increased wind speeds, corner streams are also responsible for sudden wind direction changes. The direction of the corner streams in the immediate vicinity of the building corners is determined by the direction of the facade where the flow was attached to. This direction differs from the main flow direction. Therefore, surprising effects might occur, which can be dangerous or at least unpleasant for the inhabitants and the passers-by. Also doors and windows might suffer from these effects.
2. Recreational areas around high-rise buildings should be avoided unless specific attention will be given to the design of these areas – e.g. using a canopy to block the descending flow and the frontal vortex – where the effectiveness of the specific design features is to be ascertained by the use of wind tunnel or numerical modeling.

22. What is the most important error in rainfall measurements? How can it be reduced or eliminated? What is the best sampling period for rainfall measurements? Explain why.

The error that is considered most important is the wind error. It is due to the systematic deformation of the wind flow above the gauge orifice (and hence of the raindrop movements in this flow) by the presence of the gauge body itself (Fig. 5.7a). When the rain gauge is placed at a certain height above ground (between 0.5 and 1.5 m), without anyprecautions, the undermeasurement error is typically 2 to 10 percent (WMO 1996). But it is clear that as the wind error increases with wind speed (WMO 1982), it will be much higher for rain gauges positioned on building roofs and at the top of meteorological masts. Various precautions to limit this error are all based on the same goal: to make the air flow horizontal above the gauge orifice (e.g. Meteorological Office 1956, Sumner 1988, WMO 1996).

Three recommended options to achieve this are (1) ground-level gauges (i.e. rain gauges placed in a pit with their orifice level with the ground surface), (2) to build a turf wall around the gauge as specified in Fig. 5.7b, (3) to fix a shield around the rain gauge orifice. Other systematic errors - of minor importance - are the wetting loss on the internal walls of the collector, the wetting loss in the container when it is emptied, in- and out-splashing of water and evaporation from the container. The WMO (1996) provides the following estimates for maximum evaporation losses: up to 0.1-0.2 mm per day in winter and up to 0.8 mm per day in summer. Evaporation should be limited by making sure that only a small water surface is exposed, its ventilation is minimised and the water temperature is kept low by a reflective outer surface (WMO 1996).

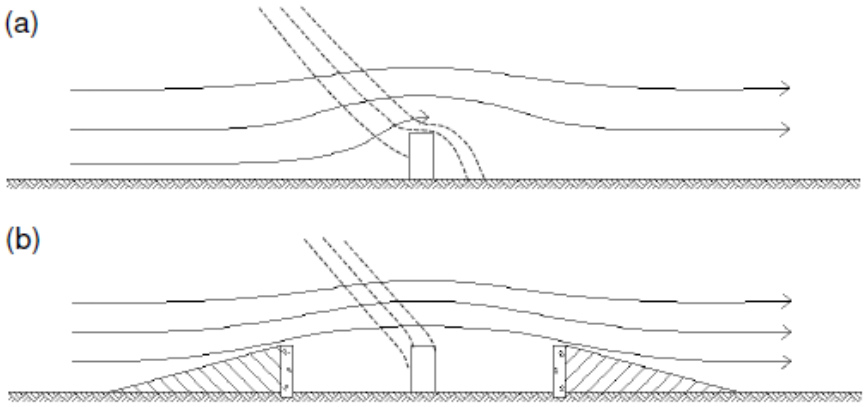


Figure 5.7. (a) Cross-section of a rain gauge exposed to wind. The gauge disturbs the wind-flow pattern and raindrops (trajectories are displayed with dashed lines) are swept over the rim of the gauge due to the sudden increase in vertical wind speed near the orifice of the gauge; (b) Cross-section of a rain gauge sheltered from wind by constructing a circular turf wall around it. The wind velocity over the orifice of the rain gauge is horizontal and as a result the trajectories of the drops are not (or only slightly) deflected.

Sampling interval for rain measurements

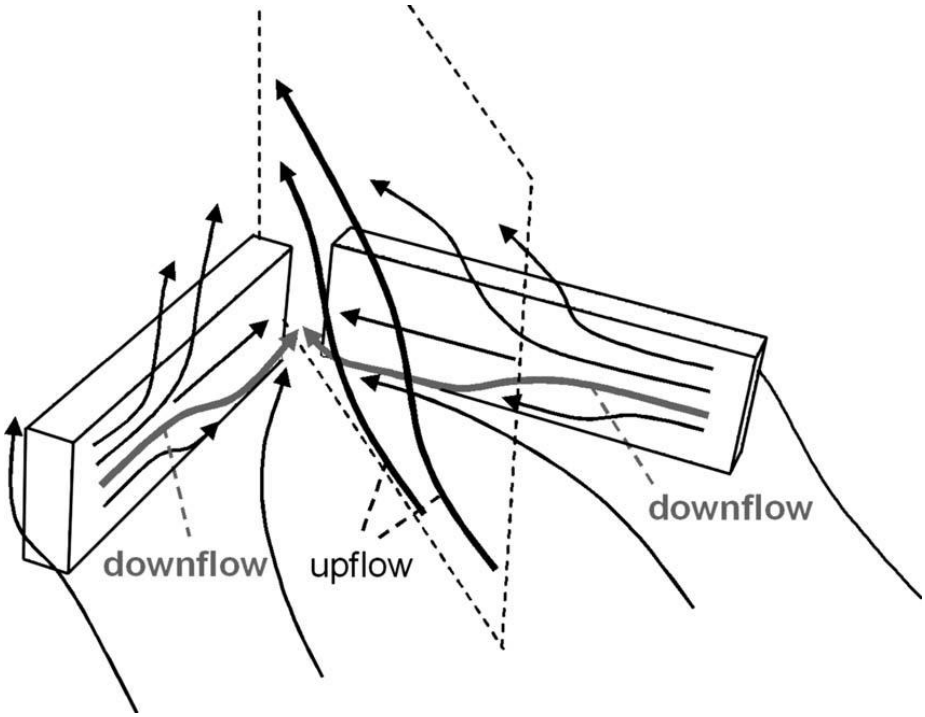
Given the extreme variability that is inherent to rain (see section 5.2.2), the minimum sampling interval for rain measurements should be selected with care. Clearly, hourly data are generally not appropriate, certainly not for the registration of short-duration showers (Sumner 1988). The shorter the time interval, the more accurate the registration of the phenomena will be. On the other hand, even the most sophisticated rain gauges are incapable of measuring instantaneous rainfall (Sumner 1988). Jones & Sims (1978) correctly state that the nature of instrumentation forces a compromise whereby instantaneous precipitation is considered to be that which occurs over a duration of one minute. Sumner (1981, 1988) mentions that due to errors in timing, local turbulence and so on, it is probably more reasonable to settle for sampling intervals of 5, 10 or 15 minutes.

23. What is the definition of the Venturi effect in urban aerodynamics? Does the Venturi effect exist in passages between perpendicular buildings in a converging arrangement? Explain why or why not.

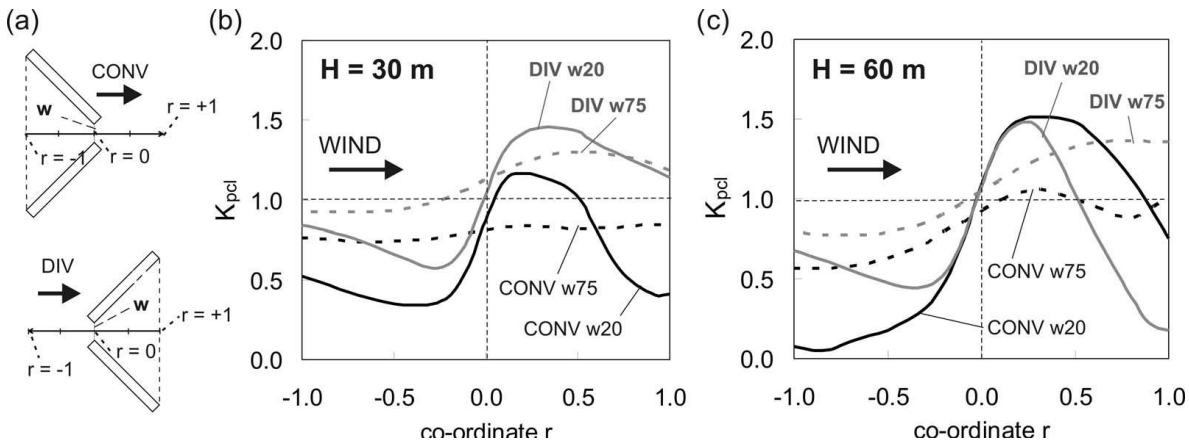
The **Venturi effect** is the reduction in fluid pressure that results when a fluid flows through a constricted section in comparison to its normal boundary flow.

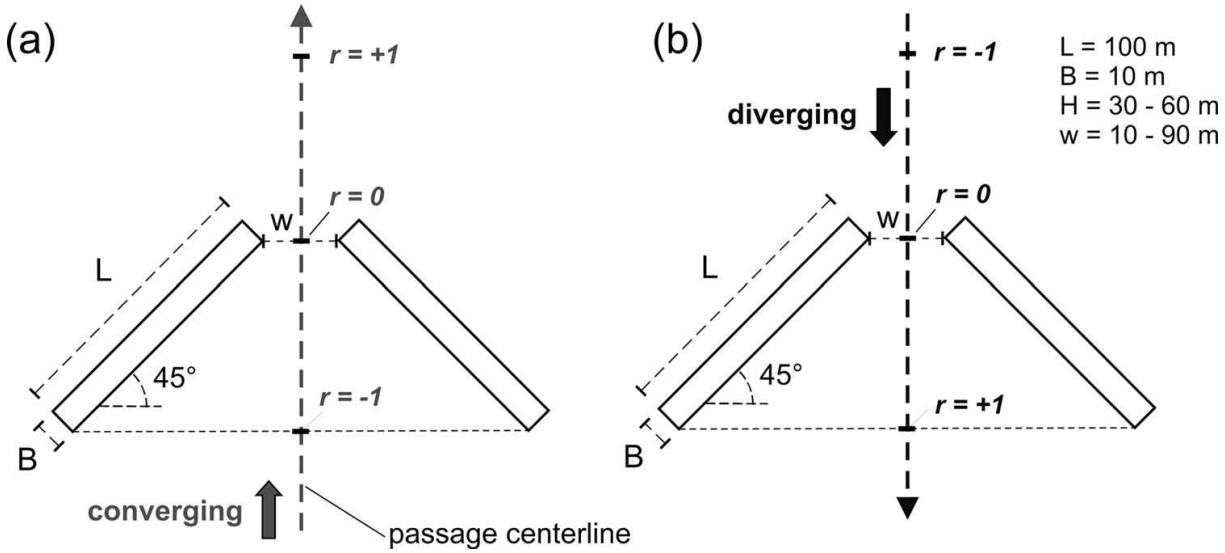
According to the laws governing fluid dynamics, a fluid's velocity must increase as it passes through a constriction to satisfy the conservation of mass, while its pressure must decrease to satisfy the conservation of energy. Thus any gain in kinetic energy a fluid may accrue due to its increased velocity through a constriction is negated by a drop in pressure. An equation for the drop in pressure due to the Venturi effect may be derived from a combination of Bernoulli's principle and the continuity equation.

1. For converging passages, there is an increase in wind speed near ground level but a decrease of horizontal wind speed in the upper part of the passage. This is due to the windblocking effect, i.e., a large amount of the oncoming air flows over and around the buildings, rather than being forced through the passage opening. As a result, the wind flow rate through the converging passages is consistently lower than the free-field flow rate.
2. The Venturi effect originally applied to flow in closed channels. This terminology cannot generally be extended to the non-confined flows in wind engineering. When the reference situation is a free-field situation, there is no increase in flow rate through the passages and strictly speaking, the so defined Venturi effect is not present.
3. Both pedestrian-level wind speed and air flow rates in the diverging passages are higher than in the converging passages, for which the wind-blocking effect is most pronounced.



It seems that when the two buildings are closer to each other the harder the wind blows between the passage, but when moving further through the passage, wind speeds will decrease if they are separated by more space. The height of the building has an negligible influence on the maximal k-factor. As shown in the figure below.



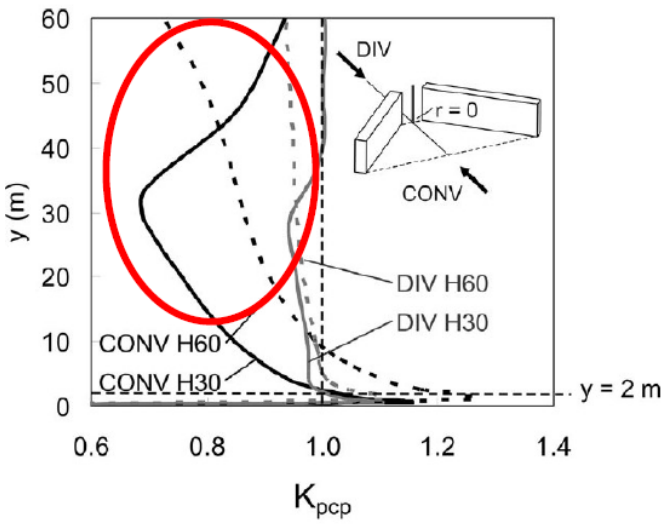


24. Concerning the wind-flow pattern through and around a converging passage between two perpendicular buildings, two facts appear to be in contradiction with each other: (1) the increase of pedestrian-level wind speed in the passage compared to the wind speed in free-field conditions and the decrease of the passage flux compared to the free-field flux. Explain, using a drawing, why this is only an apparent contradiction.

Contradiction?

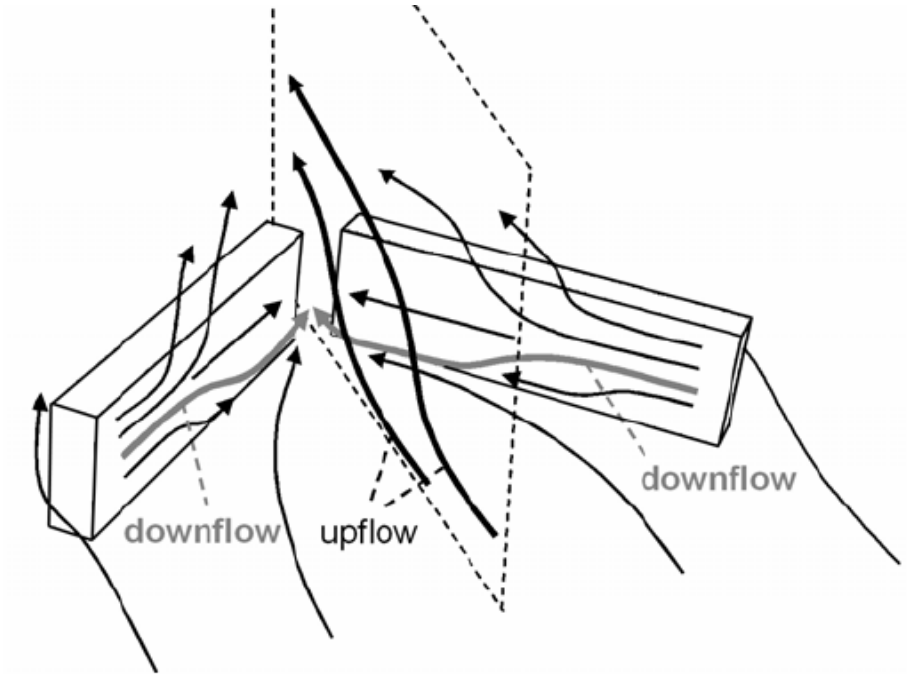
Increase in wind speed near ground level, but decrease of passage flux...

No: reason of apparent contradiction is that in converging configuration, there is a strong decrease of wind speed in upper part of the passage. The increase in wind speed is only pronounced near ground level.



K_{pcp} = amplification factor in **p**assage **c**enter **p**lane

$= U(z)/U_{free}(z)$



Kpcp is an indication of the increase / decrease of the horizontal wind speed in the passage at all heights (relative to the free field). The curves show a peak in the vicinity of the ground floor and a sharp drop in the upper part of the passage.

The peak is due to down flow, downward flow of air at the windward facades which then flows through the passage in the vicinity of the ground level. A part of the flow of wind deviated from the buildings, some flows along the wall in the downstream direction and part is deviated to the ground.

The decrease in the height with Kpcp is caused by the large amount of air which exits through the top surface (upflow). So close to the ground, the wind speed increased in the corridor, but the total passage flux decreases relative to the free field, because the wind is not the passage will go. This is an apparent contradiction because:

25. Give a definition for pedestrian wind discomfort and for pedestrian wind danger. Explain why high-rise building can cause wind discomfort and wind danger. What can be done to solve such problems, or at least to limit them?

Discomfort probability and danger probability are defined as the percentage of hours (during a year) in which the thresholds are exceeded. The maximum allowed percentage will depend on the type of human activity that is planned.

As opposed to wind comfort, wind danger can be directly related to wind effects.

For wind comfort:
$$\begin{cases} U_e = U + \sigma_u < 6 \text{ m/s} & \textbf{(22 km/h)} \\ P_{max} = 10\,\% \end{cases}$$

For wind danger:
$$\begin{cases} U + 3\sigma_u > 15 \text{ m/s} & \textbf{(54 km/h)} \\ U + 3\sigma_u > 20 \text{ m/s} & \textbf{(72 km/h)} \\ P_{max} = 1\% \end{cases}$$

High-rise building can cause wind discomfort and wind danger because natural flowing wind which flow against these buildings will be amplified on ground level, where this can be mean wind speeds two time as fast. Next to this turbulence will occur which make it harder for people to keep their balance and fall to the ground.

How buildings can cause wind discomfort?

Pedestrian wind nuisance and danger caused by the standing eddies and angle flows, and also the pressure-short (short-cutting pressure) (passages) plays a role.

The stagnation point of a building is on 2/3 of the height. high-rise buildings have tend to bring down high velocity air to the ground and the permanent vertebrae and influence currents angle.So highrise pedestrians can wind nuisance. Wind speed increases with height. There are several aspects that affect wind nuisance. This is wind speed, wind direction and dimensioning of buildings. The only aspect that we planners have effects on the dimensioning of buildings. In a building there are a number of wind currents that affect pedestrians. These are vertebrae generated perpendicular to the facade on the wind. But corner effects. First a village sketches of how the wind normally comes along and across a building. The three basic wind currents. Also explain that there is at the front of a building over pressure and under pressure at the rear. These are actually the most important effects on pedestrian level marked. these effects are not on their own very burdensome, but these can be enhanced by the shape of buildings. There are a number of building shapes that appear to overcome these effects are not extra strength.

Hole in building

In front of the building the wind. This creates a pressure on the front and a rear pressure. This would like the wind from the front to the back flow because the pressure must be reduced. Normally (without hole) this would be through the corners of the building, and the building. Now there is a hole in the building and wants the wind, so through it. This creates large movements in the hole and it does so dangerous situations.

Two separate buildings next to each other

The wind blows as it might blow to individual buildings. Because they cover each other strengthens this effect. The closer the greater this effect. The two distinct zones are a big area. If more buildings apart, people will walk in the middle where the angular effects are less. When the buildings closer together, people can not avoid these corner effects and thus it seems the wind speed increases when buildings closer to each other. This is not the case.

Two separate buildings at an angle to each other

This is actually somewhat the same situation as case 2 only the effect on an otherwise strengthened. Now there on the side with an area under pressure. And thus, between the buildings will create a lot of wind currents. When the buildings are closer together, the effect decrease. This is because the total width of the building that is lower. This will reduce the wind picked up and there will be less pressure and under pressure.

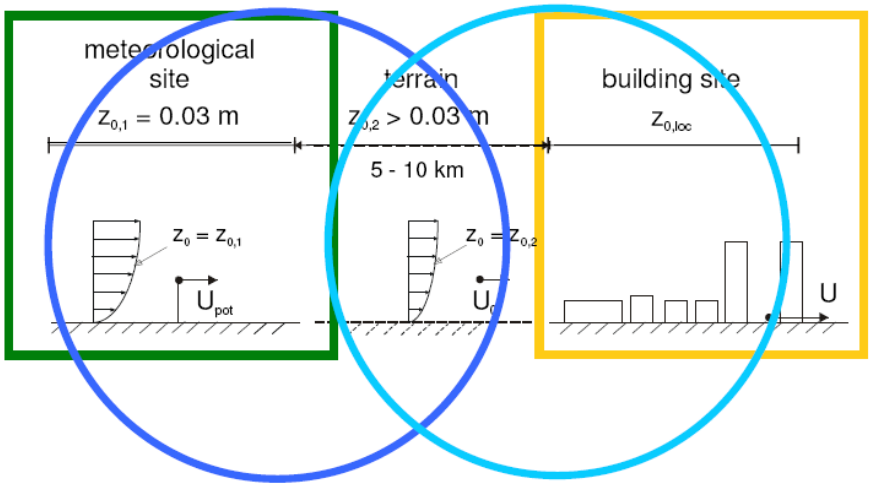
Solution

To solve or reduce the problems of wind discomfort and danger, it is important not turn doors and passageways near building corners, no passages through buildings (pressure short circuit) and do not pass-roads leading through narrow passages between buildings. Other possibilities to avoid problems are canopies, balconies and stage-shaped extensions. The last option is the vegetation in front of the building. screens and fences are not a good solution.

26. Explain the three parts of information that are needed to perform a wind comfort study. What are the advantages and disadvantages of using wind tunnel modelling or CFD (RANS and k-ε model) for such studies?

Needed:

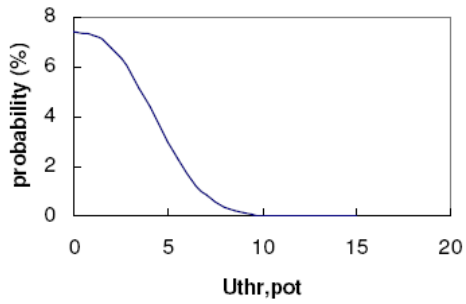
1) Statistical meteorological data
2) Aerodynamic information
3) A *suitable* comfort criterion



Statistical meteorological data

- typically 30 years
- potential wind speed and wind direction (y_0 = 0.03 m)
- Weibull distribution (fitting the parameters)

$$P_\theta\left(U_{pot} > U_{THR,pot}\right) = 100 \cdot A(\theta) \exp\left[-\left(\frac{U_{THR,pot}}{c(\theta)}\right)^{k(\theta)}\right]$$



A suitable comfort criterion

- Many criteria exist
- Generally, no experimental basis, contrasting criteria

Based on the comparison work and after adding appropriate modifications:

For wind comfort:

$$\begin{cases} U_e = U + \sigma_u < 6 \text{ m/s} \\ P_{max} = 10\,\% \end{cases}$$

(22 km/h)

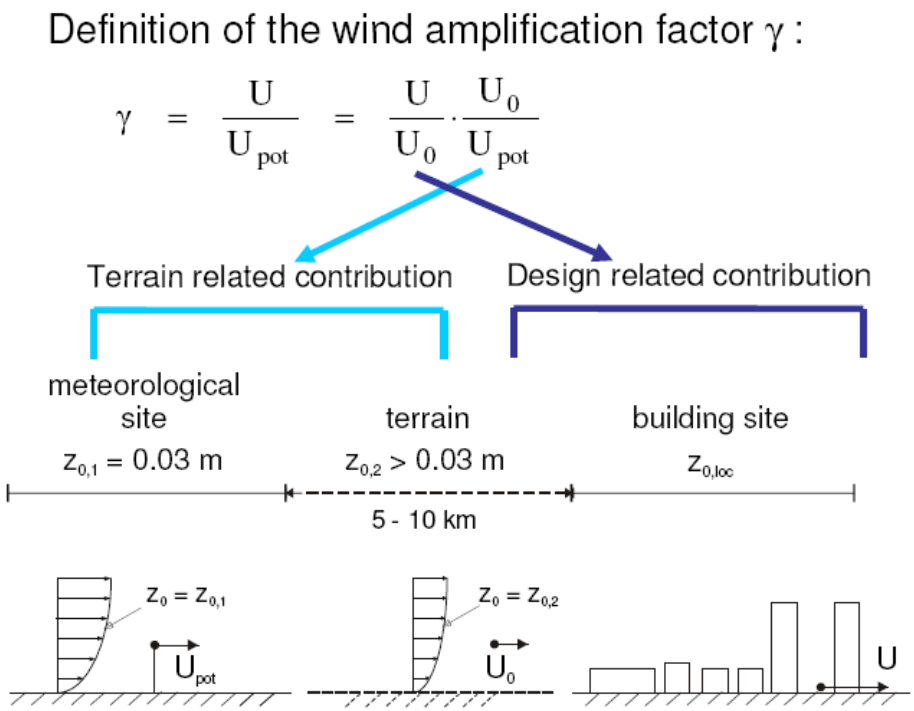
For wind danger:

$$\begin{cases} U + 3\sigma_u > 15 \text{ m/s} \\ U + 3\sigma_u > 20 \text{ m/s} \\ P_{max} = 1\% \end{cases}$$

(54 km/h)
(72 km/h)

Aerodynamic information

Definition of the wind amplification factor γ:



Advantages and disadvantages of using wind tunnel modelling vs CFD

- Wind tunnels are almost always only used by professionals where, CFD is open to everyone.

Wind tunnel

Advantages:

- Full control over boundary conditions
- Fast and efficient point and planar measurements
- Generally high accuracy
- Qualified personnel

Disadvantages:

- Scaling / similarity
- No whole-flow field data

CFD

Advantages:

- Full control over boundary conditions
- Whole-flow field data
- No scaling/similarity issues

Disadvantages:

- Time consuming (geometry and grid generation, solution procedure)
- Accuracy and reliability are main concerns
- Easy access: “everybody does CFD”

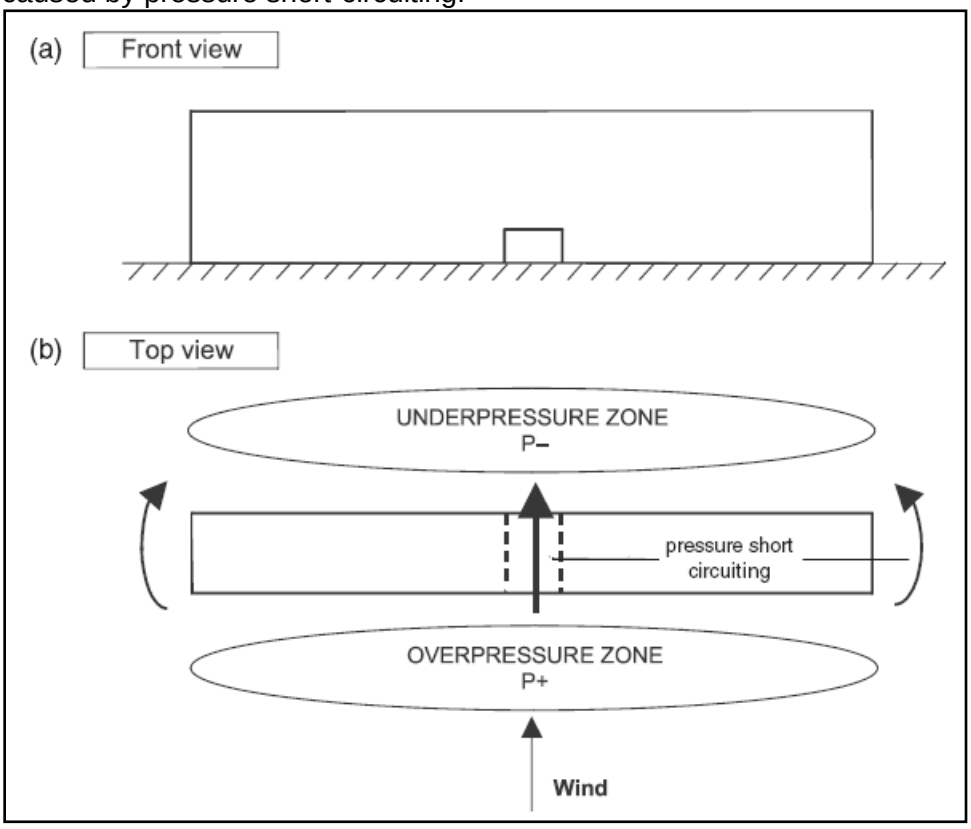
27. Which three typical building configurations will almost always cause wind comfort problems? Provide a drawing and an explanation of how these configurations cause high wind speed at pedestrian level. What can be done to limit wind discomfort problems in each of these cases?

Three building configurations that will most likely cause wind comfort problems:

- **Passage through a building**
- **Passage between parallel buildings**
- **Passage between shifted buildings**

Passage through a building

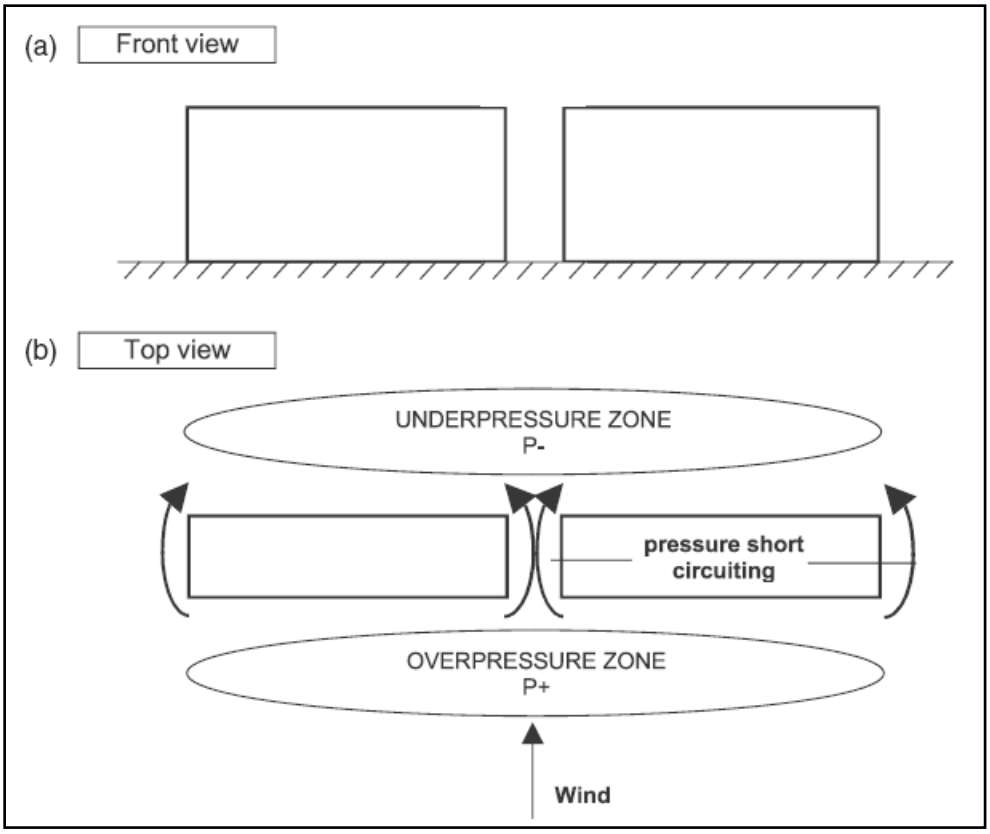
Schematic representation of pedestrian level wind flow for a building with a through-passage. The flow through the passage is caused by pressure short-circuiting between windward (overpressure) and leeward (underpressure) facade. In fact, the corner streams are also caused by pressure short-circuiting.



Wind is perpendicular to the building. An overpressure zone is created at the front side of the building, and underpressure at backside. Wind will want to flow from the overpressure to underpressure zone, to decrease the underpressure. The pressure short cut will in this case happen by flowing through the passage in the building and around the building, which creates high wind velocities and these points.

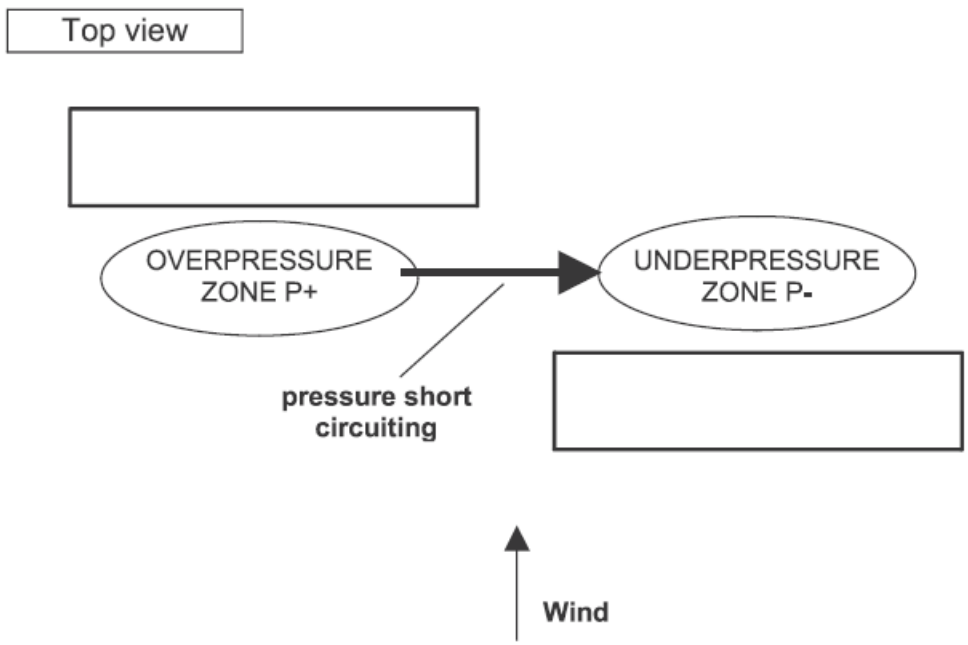
Passage between parallel buildings

Schematic representation of pedestrian level wind flow for two parallel buildings with a passage in between. Pressure short-circuiting between windward and leeward façade contributes to the flow between the buildings and the flow around the corners.



Wind blows normally as it does with normal individual building, but because they are situated next to each other, the short circuiting effect increases. The closer they are next to each other the higher the wind speeds will become. Dependent on the gap between the 2 buildings, wind speed will increase with a smaller gap, larger façade surface. Opposite can be said for a larger gap, which increases wind speed through the passage.

<u>Passage between shifted buildings</u>
Schematic representation of wind flow for two parallel buildings shifted towards each other. The transverse flow between the buildings is caused by pressure short-circuiting between the overpressure zone in front of the windward facade of one building and the underpressure zone behind the leeward facade of the other.



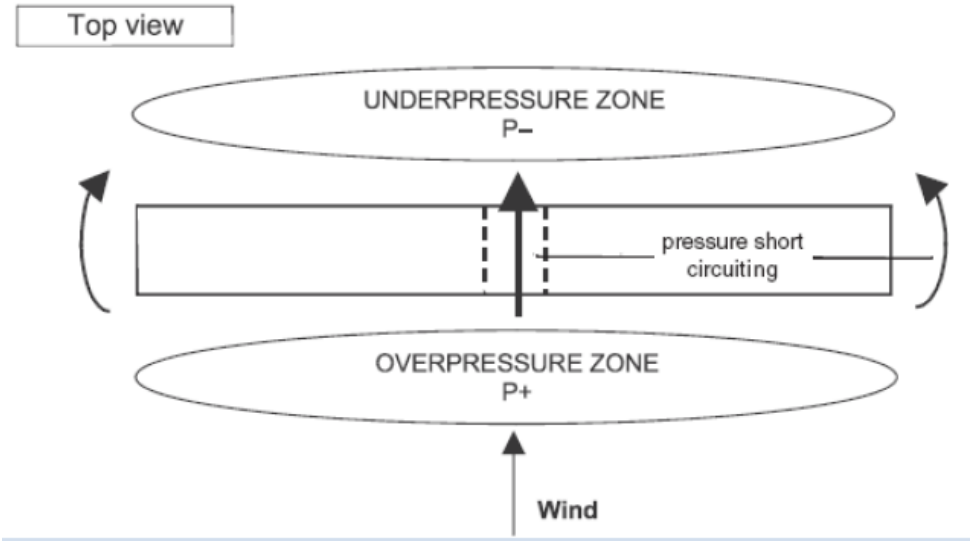
This situation is actually the same as situation 2, the difference is that the effect gets stronger in a different way. Between the two buildings, on the windward and leeward façade severe pressure short circuiting is created by the incoming wind. Here the height of wind speeds are dependent on the situation, a decreased distance will decrease the effect, where a larger gap increases the total surface and increases the pressure zones and wind speeds.

Solutions voor situation 1 and 2
Screens in the opening
Designing a tube through the building, without creating the venturi-effect. It should not increase the wind speed through the tube
Building a canopy
Close the gap
Balconies
Podium-shaped extensions
Vegetation

Solutions for situation 3
Screens in the opening
Close the gap
Move the buildings closer to each other
Balconies
Podium-shaped extensions
Vegetation

28. Figure 2 shows part of the new bus and railway station building in Leuven. People waiting for the bus at the bus stop in the opening below the building often complain about wind discomfort. Explain why this configuration can cause wind discomfort. What can be done about this in the present situation (without destroying the building). How would you design this building, in such a way that the busses can ride through the openings and that there is no pedestrian wind discomfort for passengers that are waiting?

Figure 2 shows a part of the new bus and train station building in Leuven. People waiting for the bus at the bus stop in the hole below the building often complain about problems with the wind.
In this configuration, the wind nuisance can come from pressure short-circuiting.
corner effect



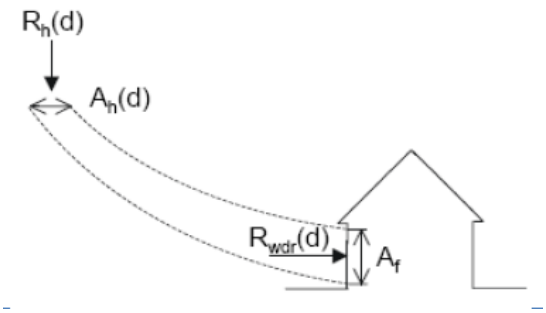
1. In the current situation, a solution can be to reach airtight tubes passage which ultimately outside the over-and under pressure zones.
2. is to close the "columns" in the passage (too little waiting rooms with sliding doors), so pedestrians can wait out the wind. Buses can drive by the wind, but it can handle.
3. Another option is to put screens in the passage to increase the flow resistance. But put the screens must be careful, because pedestrians have walk, buses have to drive, the view should not be blocked, etc.
4. in my opinion the best solution is to close the whole passage. With the use of sliding doors, the buses through the passage and pedestrians can wait "inside" out of the wind.

29. Briefly explain the five steps that are used for CFD simulation of wind-driven rain on buildings. Explain the third step in detail, including the relevant equation. On which assumption is this equation based?

1. The steady-state wind-flow pattern around the building is calculated using a CFD code.

2. Raindrop trajectories are obtained by injecting raindrops of different sizes in the calculated wind-flow pattern and by solving their equations of motion.
3. The specific catch ratio is determined based on the configuration of the calculated raindrop trajectories.
4. The catch ratio is calculated from the specific catch ratio and from the raindrop-size distribution.
5. From the data in the previous step, catch-ratio charts are constructed for different zones (positions) at the building facade. The experimental data record of reference wind speed, wind direction and horizontal rainfall intensity for a given rain event is combined with the appropriate catch-ratio charts to determine the corresponding spatial and temporal distribution of WDR on the building facade.

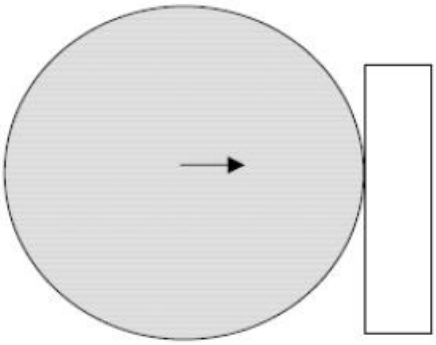
$$\mathbf{d\ (d)}\ \frac{\mathbf{Rwrd\ (d)}}{\mathbf{Rh\ (d)}} = \frac{\mathbf{Ah\ (d)}}{\mathbf{AF}}$$

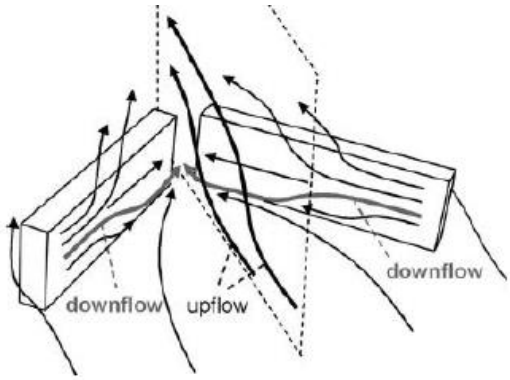


AF = surface area of the zone on the facade of a building Ah (d) = surface area of the horizontal plane defined by the injection positions of the raindrops from the diameter d ends at the corner points of AF.
- The calculation of the specific catch ratio is carried out for a number of areas on the facade of a building. For each zone, the same procedure is applied. - In the steady-state wind-flow pattern, raindrop trajectories of diameter d ends at the corner points of the area forms a stream tube. - Conservation of mass for the raindrops in the stream tube is ηd be expressed in terms of areas. The plane Ah (d) is located in the upstream undisturbed flow (Rwdr = wind-driven rain intensity (mm / h) and Rh = horizontal rainfall intensity (mm / h)

30. What is the wind-blocking effect for an isolated (single-standing) building? How can this influence the wind-driven rain exposure of a building?

Wind-blocking effect for an isolated building:
the disturbance of the wind-flow pattern by the presence of the building and the associated decrease of the upstream-stream wise wind velocity component in the vicinity of the building **(wind speed slow-down).**
Influence
The larger the area where the wind-blocking effect occurs and the stronger this effect, the more the stream wise velocity horizontal raindrop will fall, thus raindrop trajectories that are almost vertical near the lower part of the Windward side of a building. So that the wind-blocking effect is most pronounced for the high-rise building and least pronounced for the low-rise cubic building.
- In an experiment, it is clear that a higher and broader build up a greater barrier to the wind-flow pattern, which in turn can cause a lower WDR represents exposure, even at the top of the facade (experiments were carried out, without any other obstructions).
- (As an example the influence is explained. Simulation results showed a lower stream wise wind speed values in the presence of a sloping roof module is in front of a flat roof module. So a higher pitched roof module presents a greater barrier to the upstream wind flow field. As a result, a much more important stream wise wind speed slow down and thus would receive less WDR).





31. Mention at least 5 typical characteristics of the indoor air flow that have a direct link with modelling of the indoor environment in CFD? Why can these characteristics make modelling of the indoor environment in CFD more complex?

Typical characteristics

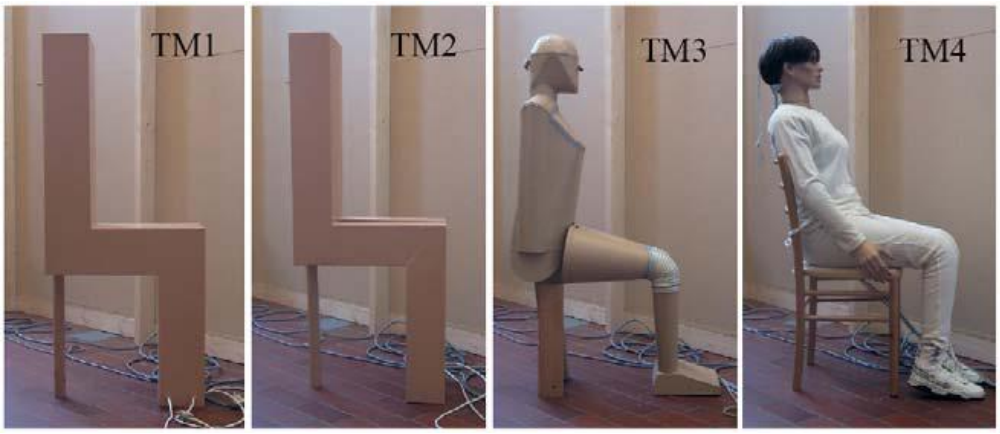
As indicated earlier, application of CFD for the indoor environment has several specific aspects compared to the general field of applications for CFD, such as aerospace and process industry and the outdoor environment. Some typical characteristics of the indoor environment would include:

- Geometry generally less complex,
- Internal obstructions/details can change this drastically,
- Generally non-isothermal situations,
- No steady-state (but assumed quasi…),
- Complex boundary conditions (supply, exhaust, heat exchange),
- Relative low velocities (> turbulence).

Complexity

-The internal details of the geometry can be very complex;
-the non-steady-state conditions must be predicted;
-the boundary conditions are complex;
-every little difference can change the solutions
-and the turbulence must be very carefully modeled because of the low speeds.

32. When modelling an indoor air flow problem, which virtual thermal manikin geometry you would prefer if:
i. It relates to an integrated performance indicator for the room under investigation?
ii. It relates to investigation of the function of a personal ventilation system?
Explain why.

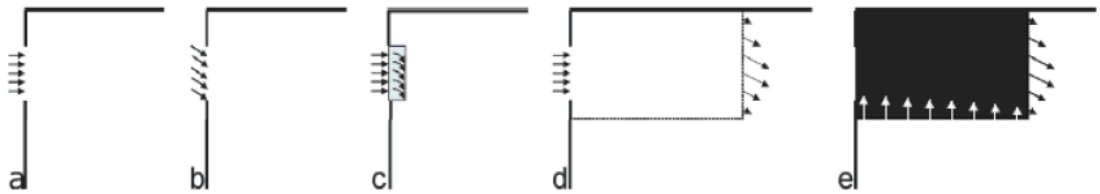


There are no general guidelines on how complex or real must have a minimal model. This depends strongly on the kind of research that is carried out.

i. In case of an integrated performance indicator for the room under investigation I would prefer to use thermal manikin geometry 1, because the only thing you are investigating is the room, excluding its users (or the users not being the main subject).

ii. If it relates to investigating the function of a personal ventilation system, it will have to take into account a realistic shape of the human body, that is TM4. The separation of the legs would give more accurate results because the air flow around the body has a big influence.

33. Which approaches are available to model a supply grille for a room? Which approach would you choose to model the supply device shown in the figure and explain this approach?



- Detailed modeling of the diffuser most realistic portrayal of the supply

A Simple Approach 1——inlet is replaced by an effective feed. The mass flow rate into space remains the same.

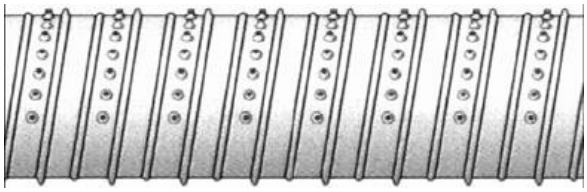
B - Simple approach 2——effective supply equal to real supply. Speed and direction of air flow included in the boundary conditions of the grid for a more realistic view

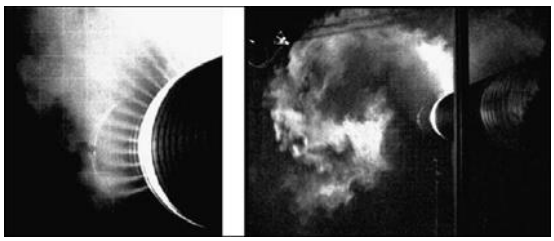
C Momentum method——real feed plane in this method is equal to the level modeled in the CFD model. This is useful for the gratings(光栅) with a small inlet area relative to the entire object.

D - Prescribed velocity method——may be comparable with the Simple Approach, where a velocity profile is modeled, but with the velocity profile, it is fixed on a certain number of positions for the inlet. The flow field adjusts to (take them as type of boundary conditions).

E-Box method——kind of extended version of the "Prescribed velocity method. It ignores the real supply. Instead of around the inlet, the black box has boundary velocities fixated on a number of points.

For the above feeding device, the momentum method is the most suitable one. The model is useful for supply grilles with a small effective area compared to the actual dimensions of the grille. Modeling of the effective area would lead to a possible unrealistic effect on the downstream flow field. Modeling of the actual supply area would underestimate the momentum that is introduced. By adding the momentum in a volume in front of the supply this can be corrected. Supply conditions agree with the actual mass flow rate, temperature and contaminant concentration.





34. For the assessment of which general indoor environmental performance indicators in the design phase use of CFD would be required (at least 5)? What can make the assessment critical?

Several indoor environmental performance indicators are;

- Temperature
- Air velocity
- Predicted Mean Vote (PMV)
- Draught (PD)
- Temperature gradient
- Ventilation efficiency
- Air change efficiency (εa)

Performance indicators and related physical attributes and target values can put stress on the accuracy of numerical simulation (and experimental results). So the sensitivity of these performance indicators is very important. And for CFD modeling, the distinction is very critical for the assessment, which is based on Geometry definition and discretization, Boundary conditions and Turbulence modeling.

35. Which force makes solving indoor air flow problems generally a tedious task? How is this generally approximated for the conditions generally present in the indoor environment? What are the assumptions in this approximation?

Thermal or buoyancy forces The effect of density differences.
Generally the Boussinesq approximation is applied leading to the following assumptions.
- Density is constant only where thermal forces result from it.
- All fluid properties are constant, at a characteristic temperature T0.
- Viscous dissipation is neglected.
- The density in the buoyancy term is linear.

36. For an operating theatre environment, which performance indicator is most important? What is the best approach to minimize the value for this performance indicator? What other indicators you can think of in relation to the operating theatre that would require the use of CFD?

Air quality (air pollution as low as possible), as exemplified by a target of <10 CFU / m^3.
<u>The best way to value for this performance indicator to minimize</u>
- By reducing pollution sources and by having control of the air flow.
- The latter can be achieved with a new type of ventilation system: laminar downflow system.

Other indicators can include:
1. hypothermia (with a simulation of the human body)
2. thermal comfort (ie temperatures
3. temperature gradients
4. design, air velocities) and
5. in the air change efficiency.

37. Which additional equation is solved in case of the presented application example for the operating theatre? Which two alternative approaches would have been possible and what is the main difference between the three approaches?

The scalar conservation equation in the passive scalar approach (without assumptions on the effect of the particle weight on the concentration distribution).

- **Lagrange approach** – uses particle trajectory equations to determine the path of each individual particle. Based on a large number of path lines, the contaminant concentration can be derived.
- **Euler approach** – applies the scalar conservation equation, but includes influence of the particle settling velocity on the contamination concentration.

The difference is how finely the approach to simulate (accuracy). Lagrange looks at each particle, Euler browsing this less close / accurate, but the effect on the particle settling velocity and passive scalar approach is the same as Euler, but without the effect of the particle size.

38. Which two main ways of modeling the combustion process in a fire are often used? Explain the difference and present an example application where you would prefer one over the other.

Combustion process – This process causes the production of heat and smoke. Two main approaches are available:

- **Volumetric heat source model**; does not predict the release of heat and smoke in the flame but the transport away from it. Quantities of heat and smoke are specified and assumed distributed uniformly in a given flame volume.
- **Combustion model**; tries to predict the chemical reactions in a flame. Overall quantity of heat release and area of fire have to be specified. However, non-uniform distribution of heat in the flame and influence of local flow is taken into account.

The **heat source model** approach is most simple and is generally applied for fire in large spaces, (for example; car parks, big atriums, shopping malls). This model is sensitive to the flame volume and, e.g. under predicts the local temperatures at the start of a fire.
The **combustion model** is to be preferred for smaller spaces and fires that are affected by the boundary conditions.

39. For the reference study that was presented the Heselden fire case has been used. Which best practice would you propose for a CFD study for this case and why?

The goal for the Heselden Case was to investigate the temperature and velocity in the “room” related to the height of the room, due to a fire. In the CFD that I should propose for this case I would keep the same goals. (It’s even possible to compare results with the measurements / other results). The best practice takes different aspects of modeling into account:
- Fire source (combustion process/heat and smoke release): The heat source model approach is chosen best here, because it’s a large space where the goal is to investigate the temperature in the space. This approach predicts the transport of heat and smoke away from the flame. The heat re-lease has to be specified by the user and relates to the burning material available (a time-dependant solution and simulation is important). The smoke release has to be specified by the user too. The simple approach combines smoke release with the heat release via an experimentally determined yield factor. This approach is the most practical one.
- Smoke movement: The smoke distribution is determined by solving the transport equations for a passive scalar. Also the Lagrangian approach is possible to track the movement of smoke particles.
- Room outside: It’s better to not leave the room outside out of the CFD study. - Boundary conditions wall/openings: The boundary conditions are comparable to other simulations. So an inlet, outlet, walls, etc. have to be specified. A transient simulation will be required, so also transient boundary conditions. For openings it’s better to prescribe the external pressure at

0Pa. The walls can be given a temperature and thickness, that’s preferable over adiabatic walls.

- Grid (2D or 3D): The choice between a 2D and 3D grid depends on the kind of study. Over here I should propose a 3D grid so the temperature and velocity distribution in a room can be checked everywhere. The results must be shown in 2D because the reference case shows the 2D flow.
- Grid study: A good 3D grid must have at least 100kcells. If one chooses a 2D grid at least

5000cells are required. A grid study must always be performed; the solutions must be grid-independent. I should propose to perform the simulation three times with a finer grid, it's ok when the solutions stay more or less the same.
- Turbulence modeling: LES can be used when the right CFD-code is available. I should propose the use of the RANS model, with at least two-equation models (like the standard k-ε model). The buoyancy effect must be taken account for in the turbulence equations.
- Radiation model: In this study I should use a radiation model because other aspects of the modeling can capture the flow behavior adequately. So the radiation model gives an added value. - Solver (transient/steady state): both are possible but related to heat/smoke release it's better to perform a transient simulation.
- Order scheme: a first order scheme would sometimes not be published. And it is also better to perform a simulation with at least a second order scheme, so that's what I should propose.
- Monitoring points (yes/no): these points are to check the convergence. Output the parameters like temperature and velocity to monitor points. If these parameters are not changing to an accuracy tolerance than the simulation is almost certainly good enough.
- Buoyancy: The boussinesq approximation must not be used for fire and smoke modeling because the temperature differences are much larger than for which the approximation holds. The buoyancy effect can be accounted for in the turbulence equations.

40. Describe all the practical steps that you will have to proceed when performing a CFDstudy of a well described flow problem.

1. Do the geometry
2. Give a grid of (how good, structured / uniform)
3. Give the boundary conditions
4. Solving transport equations by:
5. Radiation Model Select
6. Turbulence model choice
7. Upward pressure model choice
8. Compute
9. Validate