Introduction to CFD

February 2016

Content

- Basic aspects and physical properties of fluids
- Basic understanding of viscosity and turbulence
- What is CFD and basic workflow?
- Introduction to the instantaneous 3D Navier-Stokes equations
- Basic ways to solve the Navier-Stokes equations
- Introduction to Reynolds Averaged Navier-Stokes (RANS) for incompressible fluid flow
- Discretization
- Error and uncertainty

Expected Learning Outcome

- Have an basic understanding of viscosity and turbulence
- Understand how difficult it is to describe turbulent flows
- Able to identify each term in the Navier-Stokes equations, for incompressible fluid flow
- Describe basic information about the methods used to solve Navier-Stokes equations
- Basics on discretization

Basic aspects and physical properties of fluids

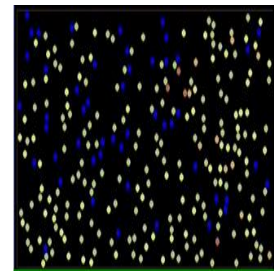
What is a Fluid?

A substance that continuously deforms under shear stress. Gases and Liquids.

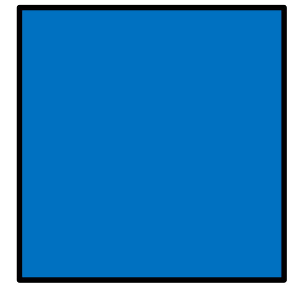
Continuum assumption.

Knudsen number $kn = \frac{l_{mfp}}{l_{ref}}$

Where, l_{mfp} is molecule mean free path and l_{ref} is a typical lenght.



Discreet molecules



Continuum assumption

Velocity
Density
Pressure
Temperature
Viscosity
Turbulence

Cup anemometer

Barometer

Viscosity

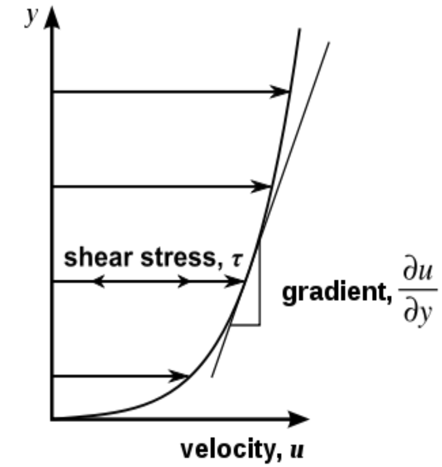
Viscosity may be understood as a measure of the fluid’s resistance to deformation by shear stress or tensile stress.

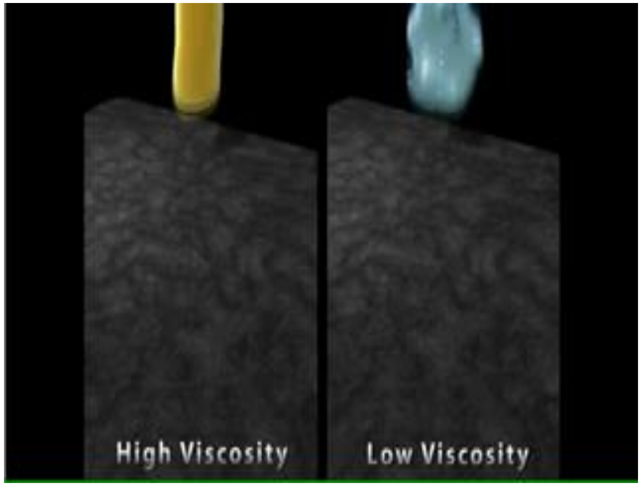
High viscosity => high resistance to deformation
Low viscosity => low resistance to deformation

Example of Absolute Viscosity at 20°C:

Air = 1.983 × 10-5 Pa.s
Water = 1.0 x 10-3 Pa.s
Olive oil = 1.0 x 10-1 Pa.s
Honey = 1.0 x 101 Pa.s

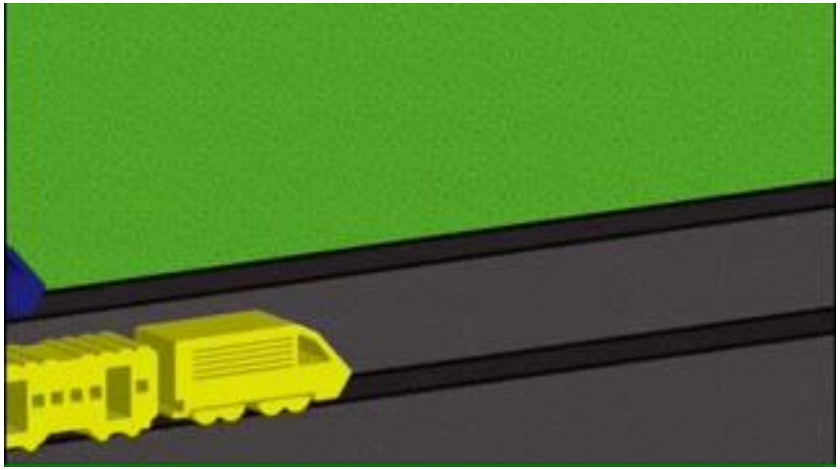
$$\tau = \mu \left(\frac{\partial u}{\partial y}\right)$$





Source: Koldora (https://www.youtube.com/watch?v=Y_48VebMTSY)

Train Analogy for Viscosity



Note: A laminar flow is assumed here.

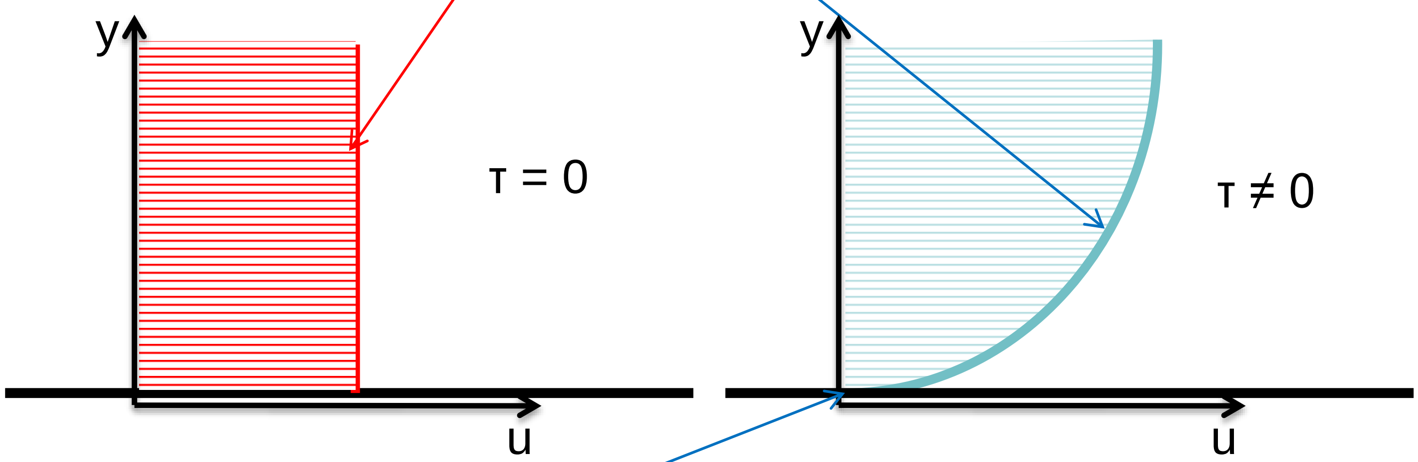
Molecular Viscosity

Slip and no-slip behavior

Wind speed profile

Inviscid flow (Ideal flow)

Viscous flow



Slip

No Slip

What do we mean by zero wind speed?

This is the reason dust accumulates on the surface of bodies even when they are in flow i.e. Blades of a fan, wind turbine blades etc.

Turbulence

Encyclopedia Britannica:

Turbulent flow, 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.

However, there is not a generally accepted definition for turbulent flows.

May Reynolds number $Re=UL/v$ define if the flow is turbulent?

- Not clear what choice of U and L in the flow should be used.
- Transition from laminar to turbulent flows does not always occur at the same or at a fix Reynolds number.
- However flows with a Reynolds number lower than 2000 is laminar and over 20 000 it is turbulent.

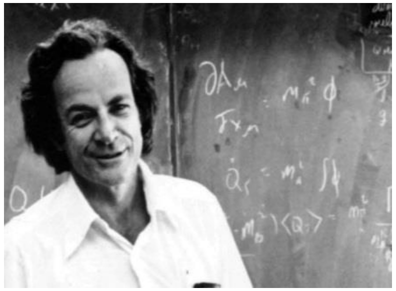
Quotes on turbulence

Richard Feynman :

referring to turbulence "the most important unsolved problem of classical physics."

Horace Lamb (1932):

"I am an old man now, and when I die and go to heaven there are two matters on which I hope for enlightenment. One is quantum electrodynamics, and the other is the turbulent motion of fluids. And about the former I am rather optimistic."



Richard Feynman, Nobel Prize-winning Physicist

Credit: MacTutor

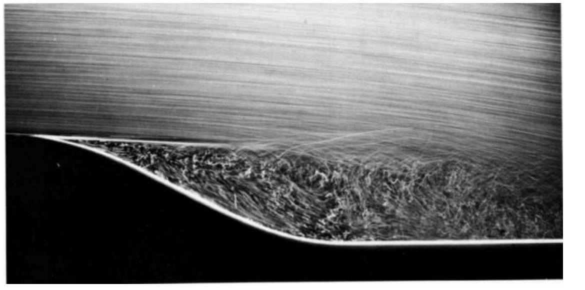
Horace Lamb
British applied mathematician

General properties of turbulent flows

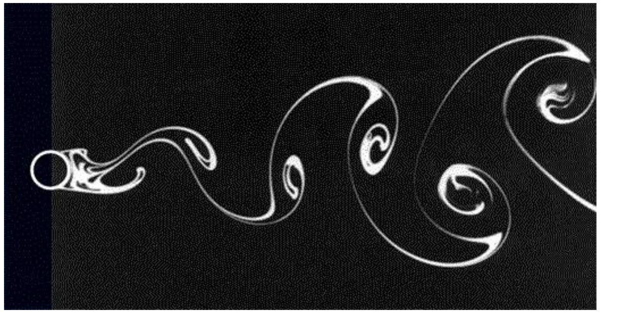
From Ferziger & Peric 2002:

- Turbulent flows are highly unsteady ("appear chaotic" random)
- They are 3-dimensional (even if mean flow is 2D)
- Large amount of vorticity present

- Conserved quantities are effectively stirred, mixed: turbulent diffusion (exchanging parcels of fluid)
- Mixing is a dissipative process (kinetic energy to thermal energy through viscosity)
- Coherent structures are present. Fluctuations occur on a broad range of length and time scales (large range of "eddies")



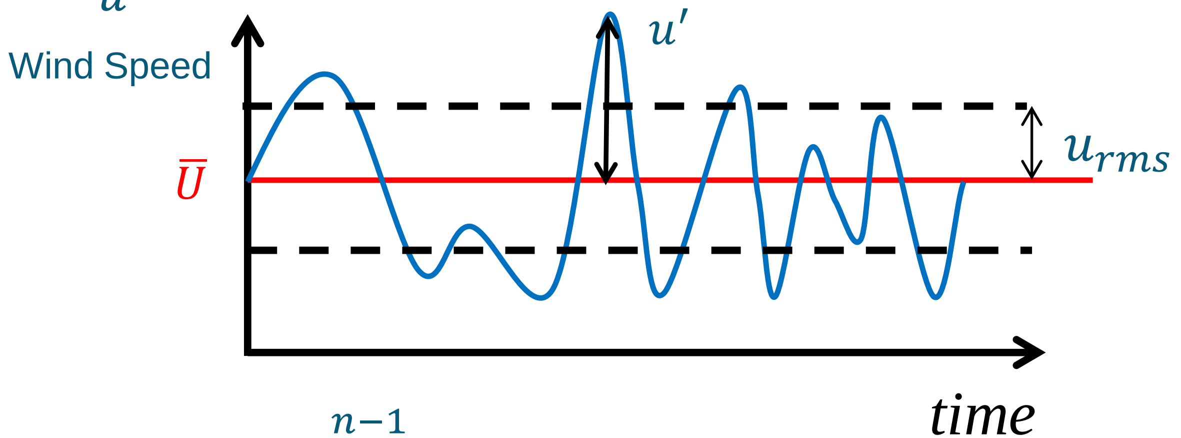
Source: Milton Van Dyke, 1982



Source: Milton Van Dyke, 1982

Turbulence Intensity

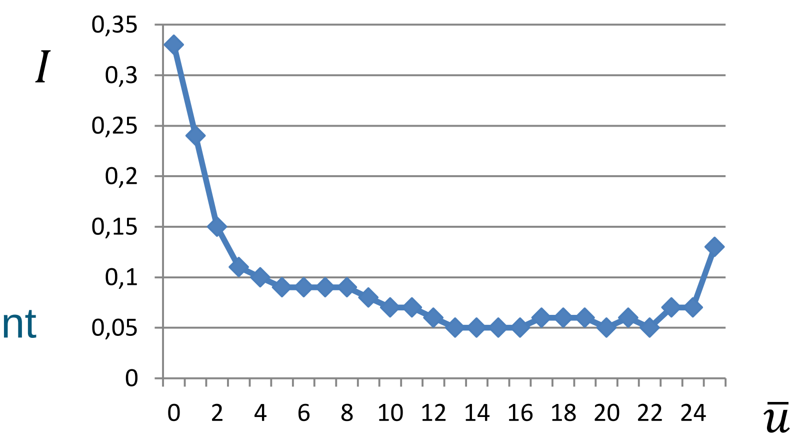
$$I = \frac{u_{rms}}{\bar{u}}$$ (Turbulence Intensity)



$$Where\ u_{rms} = \sum_{i=1}^{n-1} (\bar{U} - u_i)^2$$ (Standard Deviation)

And $u' = u_i - \bar{U}$

u_i is the instantaneous velocity, $\bar{U}$ is the mean velocity and u' the turbulent part.



CFD

What is Computational Fluid Dynamics?

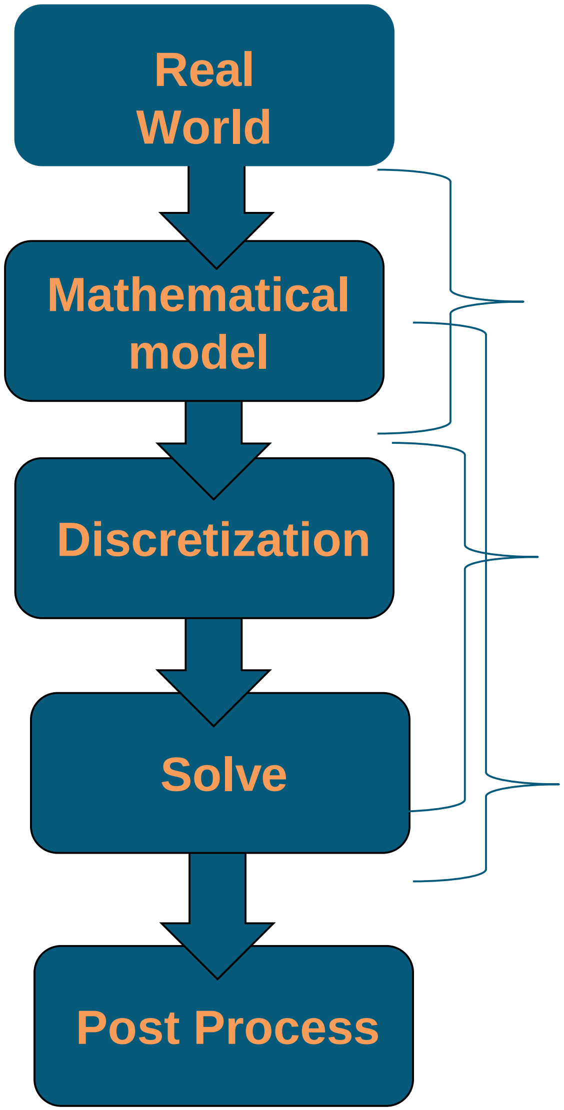
CFD is “solving fluid flow problems numerically”

"CFD is the art of replacing the integrals or the partial derivatives (as the case may be) in the Navier-Stokes equations by discretized algebraic forms, which in turn are solved to obtain numbers for the flow field values at discrete points in time and/or space.”(John D. Anderson, Jr. 1995).

General purpose CFD software packages:

PHOENICS (CHAM)
CFX , FLUENT (ANSYS)
STAR CCM+ (CD-adapco)
OpenFOAM (OpenFOAM Foundation)
and many many more

Basic CFD Workflow I



Navier Stokes, simplification assumptions etc.

Finite Volume, Finite Difference, QUICK etc.

Segregated solver, GCV, MIGAL, SIMPLE, SIMPLER, SIMPLEST etc.

Basic CFD Workflow II

Preprocessing

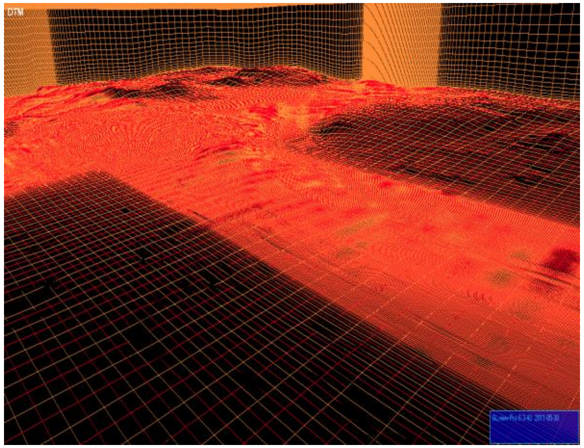
- Define computational domain: the geometry of the region of interest
- Grid generation
- Selection of the physics and chemical phenomena that need to be modelled

Solve

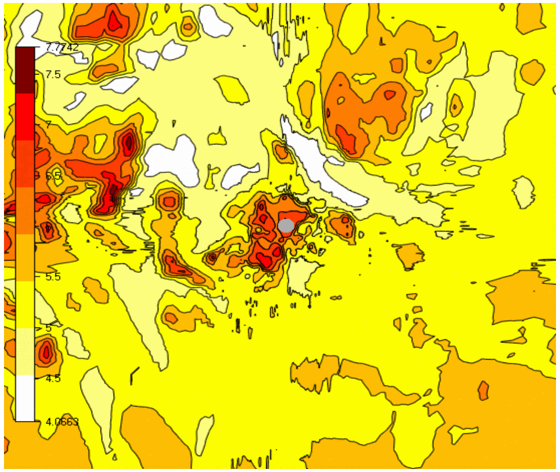
- Initialize solution and solve iteratively
- Select solution algorithm (Finite difference, finite element, finite volume and other)
- Select convergence criteria

Post processing

- Visualizing and analyzing data
- Error analysis(Verification & Validation)



3D meshing with refinement



Wind resources map

Instantaneous 3D Navier-Stokes, for incompressible fluid flow

$$\frac{\partial u_i}{\partial x_j} = 0$$

Mass conservation

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(2 \nu s_{ij})$$

Momentum conservation

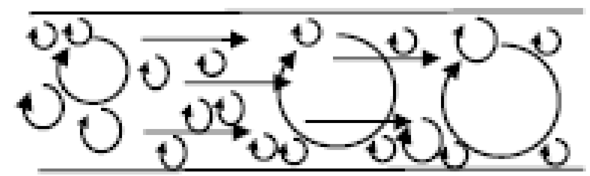
- $\frac{\partial u_i}{\partial t}$: Rate of change of u
- $u_j \frac{\partial u_i}{\partial x_j}$: Rate of flow. Convective or advective term
- $\frac{1}{\rho}\frac{\partial p}{\partial x_i}$: Pressure term
- $\frac{\partial}{\partial x_j}(2 \nu s_{ij})$: Viscous term diffusion term

$$s_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

Solving Navier-Stokes

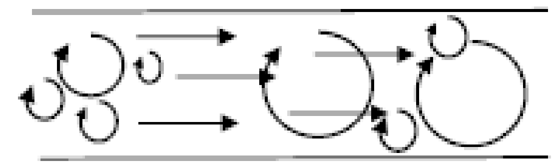
DNS: Direct Numerical Simulation

- Solve the exact Navier-Stokes equations,
- All vortices/eddies are solved, nothing is modeled,
- Large computational resources needed and very time-consuming,
- Only very simple cases are solved producing large amounts of data



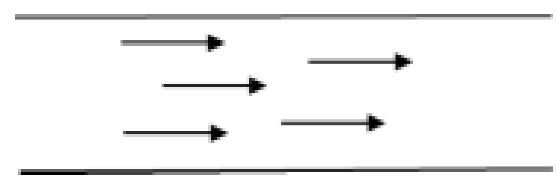
LES: Large Eddy Simulation

- Solve the filtered Navier-Stokes equations,
- Only large eddies are solved, while small eddies are modeled,
- Not exact, however less computationally demanding then DNS.



RANS: Reynolds Averaged Navier-Stokes

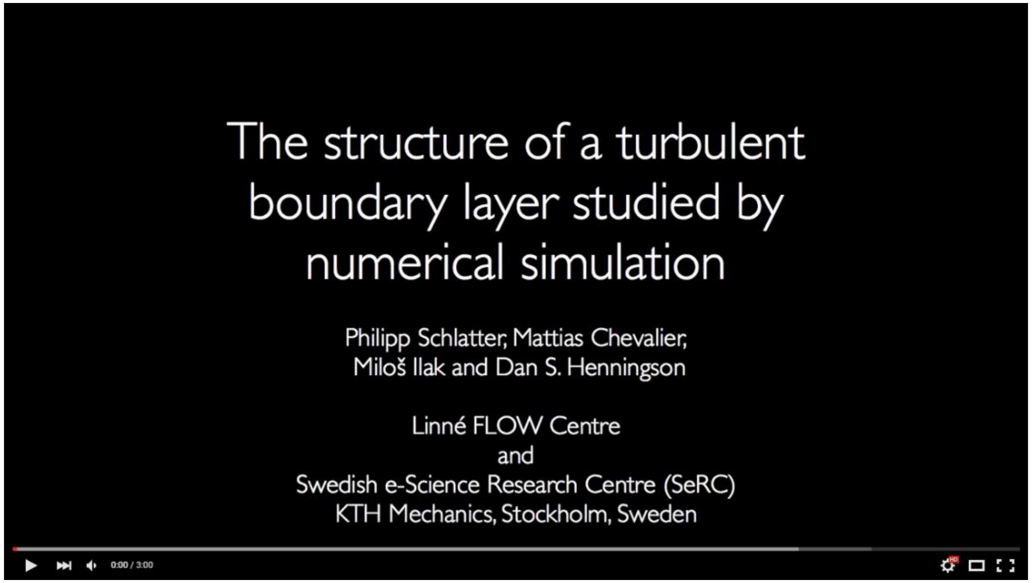
- Solve the averaged Navier-Stokes equations,
- Only the mean flow is solved, all eddies are modeled,
- Not exact, less accurate, but generally applicable,
- When solving with RANS , the effect of turbulence on the mean, flow is modeled, only the mean flow is solved.



Need Turbulence Models

Direct Numerical Simulation (DNS)

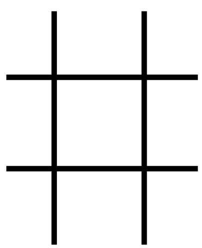
- Superior then LES and RANS.
- Extremely time consuming and very high computational resources needed.
- Used in fundamental research in turbulence.
- No need for Turbulence models.
- Example of DNS
- Turbulent Boundary Layer (KTH DNS)



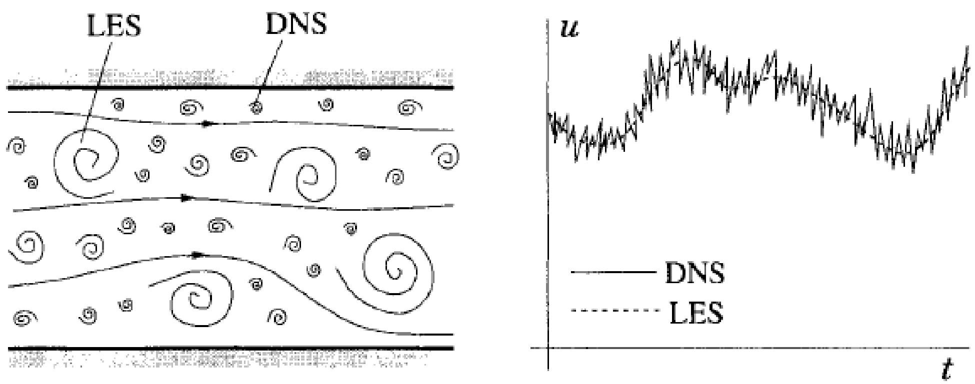
Source: Youtube (https://www.youtube.com/watch?v=4KeaAhVoPIw)

Large Eddy Simulation (LES)

- Superior then RANS when best practice guidelines are followed.
- Very time consuming, high computational resources needed.
- High level of experience needed from user.
- Spatial filtering applied. Usually the size of the grid.
- Small scale eddies accounted for by Sub Grid Scale stresses(SGS) models.
    - Smagorinsky-Lilly model
    - Dynamic SGS model
    - other



Grid Size



Ferziger & Peric 2002

Schematic representation of turbulent motion (left) and the time dependence of a velocity component at a point (right)

Reynolds Averaged Navier-Stokes (RANS)

- Time consuming, high computational resources needed but less than LES.
- High level of experience needed from user.
- Only averaged values calculated, all eddies modeled.

Turbulence models

First order (Spalart-Allmaras, K-epsilon etc.)

Further may be distinguished depending on the additional transport equations solved into:

- Zero equation: mixing length model and other.
- One equation: Spalart-Allmaras and other.
- Two equation: standard K-epsilon, RNG K-epsilon, Realizable K-epsilon, K-omega, Shear Stress Transport (SST) K-omega and other.

Second order (Reynolds Stress Models)

Originated from the work of Launder in 1975. Seven additional transport equations are used.

Instantaneous 3D Navier Stokes, for incompressible fluid flow

$$\frac{\partial u_i}{\partial x_j} = 0$$

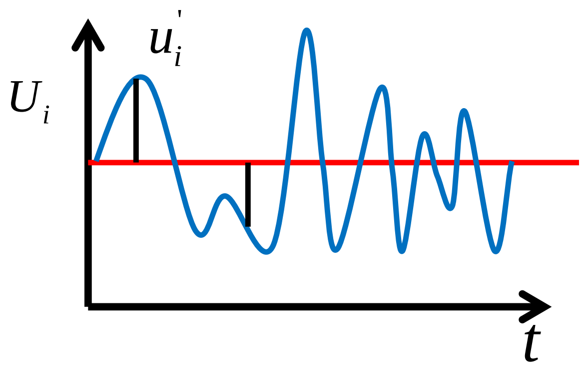
4 unknowns (u_i , p)
4 equations
Closed system

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(2 v s_{ij})$$

$$s_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

Reynolds Decomposition

$$u_i = U_i + u_i^{'}$$



Where u_i instantaneous wind speed, p the pressure, v molecular kinematic viscosity, s_{ij} is strain rate tensor, ρ density, t time, U_i mean wind speed, $u_i^{'}$ deviation from the mean wind speed.

Reynolds Averaged Navier-Stokes (RANS) for incompressible fluid flow

$$\frac{\partial U_i}{\partial x_j} = 0$$

Reynolds stresses

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}(2\nu S_{ij} - \overline{u_i' u_j'})$$

$$S_{ij} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)$$

The Reynolds Stresses represent the influence of the turbulence on the mean flow

6 new unknowns:

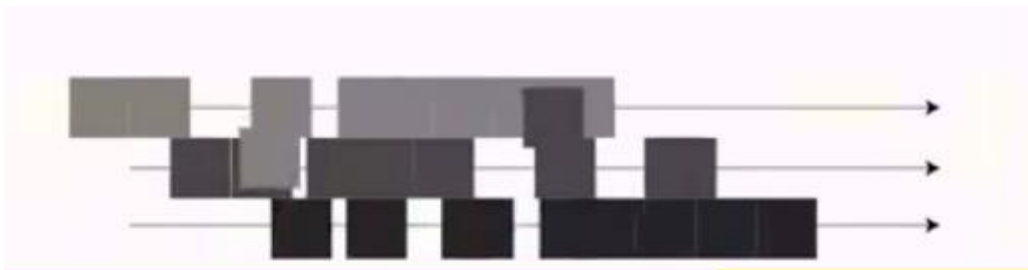
$(\overline{u'^2}, \overline{v'^2}, \overline{w'^2}, \overline{u'v'}, \overline{u'w'},$

System Unclosed

Where U_i mean wind speed, P the mean pressure, v molecular kinematic viscosity, S_{ij} is mean strain rate tensor, ρ density, t time,

Molecular vs Turbulent Viscosity

$\overline{u_i' u_j'}$ Represents effect of Turbulence on the Mean flow. May be interpreted as the momentum exchange due to exchange of **fluid parcels**. Reynolds stresses are linked to Turbulent Viscosity.



Source: Prof. dr.ir Bert Blocken

Note: This is different the molecular viscosity. Where momentum exchange is due to the exchange of **fluid molecules**.

Source: Prof. dr.ir Bert Blocken

Best Turbulence model

There is no turbulence model that is universally valid.
RSM models have more potential than two equation models. However in some flows their performance is hardly better or even worst.
Turbulence models is still a topic under reasearch.

Quote from the book of Ferziger & Peric (2002 page 306):

"Which model is best for which kind of flows (none is expected to be good for all flows)is not yet quite clear, partly due to the fact that in many attempts to answer this question numerical errors played a too important role so clear conclusions were not possible …In most workshops held so far on the subject of evaluation of turbulence models, the differences between solutions produced by different authors using supposedly the same model were as large if not larger than the differences between the results of the same author using different models."

Discretization

Commonly used discretizations methods:

- Finite Difference
- Finite Volume
- Finite Element

(Spectral method, Boundary element method, Cellular automata…)

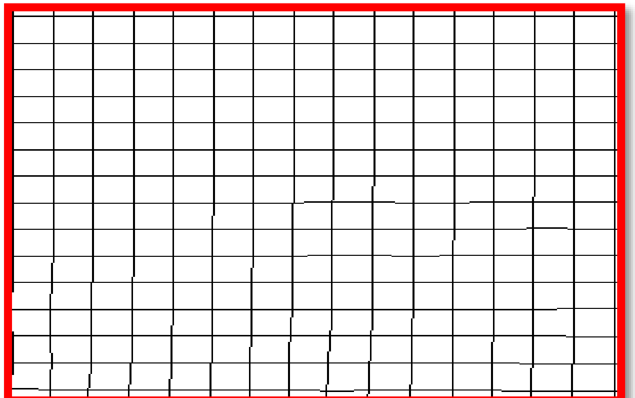
Discretization may be separated in two basic categories:

Space discretization:
- Discretize the space/domain into a finite number of points or cells.

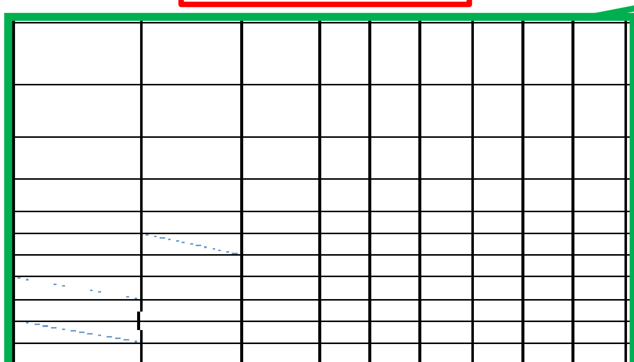
Equation discretization:
- Discretize the governing differential or integral equations into discrete algebraic expression that may be solved numerally at the grid points.

Space Discretization

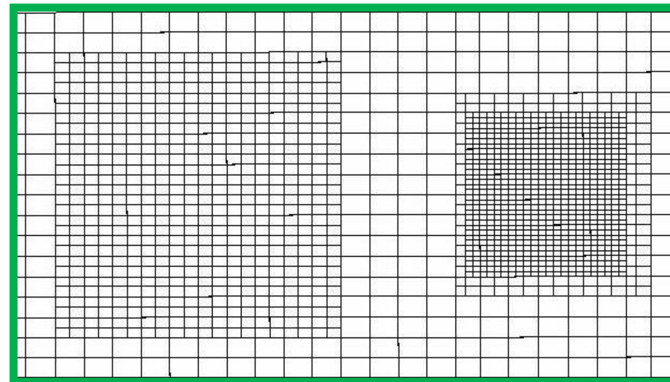
Structured Grid
(Regular topology)



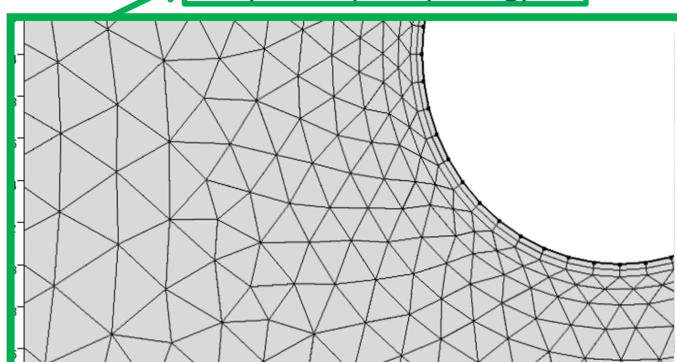
Uniform grid
(Equal spacing)



Block Structured & Unstructured Grid
(Irregular Topology)



Non Uniform grid
(Non equal spacing)



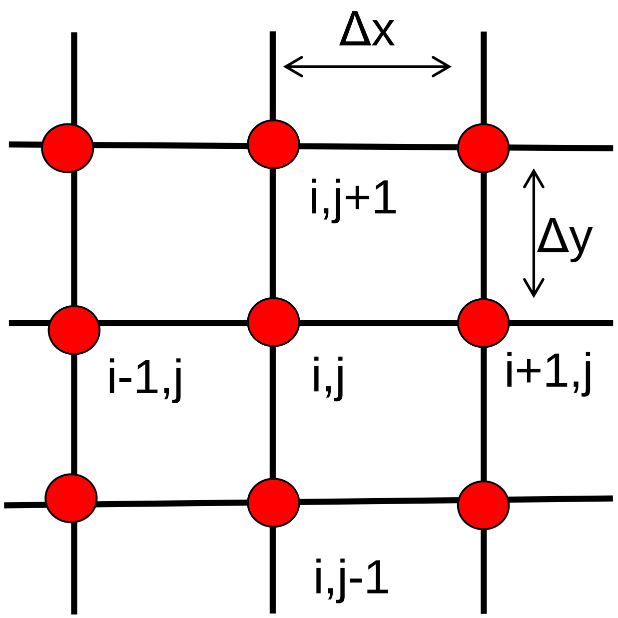
Discretization equations

Upwind differencing scheme

$$\left[\frac{\partial u}{\partial x}\right]_{i,j} = \frac{u_{i+1,j} - u_{i,j}}{\Delta x} + \vartheta(\Delta x)$$

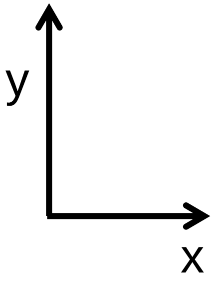
Central differencing scheme

$$\left[\frac{\partial u}{\partial x}\right]_{i,j} = \frac{u_{i+1,j} - u_{i-1,j}}{2\Delta x} + \vartheta(\Delta x^2)$$



Hybrid differencing scheme

Based on Upwind and Central differencing scheme (Spalding 1972)



Power law scheme

QUICK scheme

Errors and Uncertainty

Verification

Cause of Error:

- Numerical errors
  - Round off error
  - Iterative error
  - Discretization error
- Coding errors
  - Mistake or “bug” in the software
- User errors

“ Solving the equations right”
(Roache 1998)

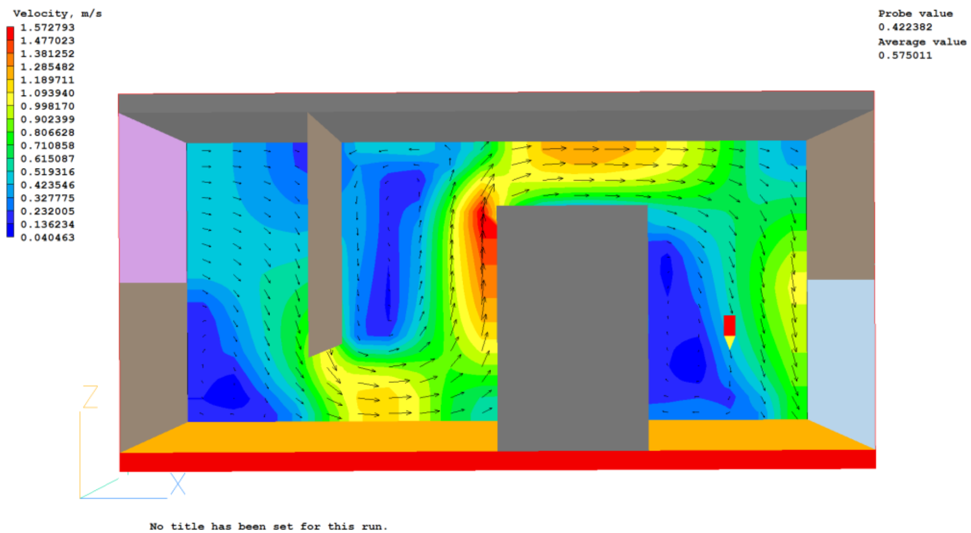
Validation

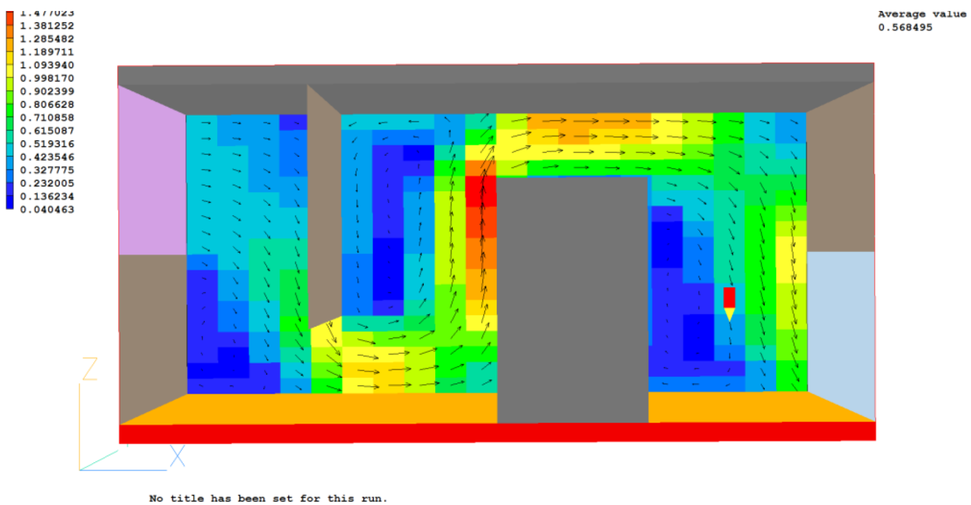
Cause of Uncertainty:

- Input uncertainty
  - Inaccuracy due to lack of information
  - Approximation of geometry, boundary conditions etc.
- Physical model uncertainty
  - Inaccurate representation of physical or chemical process (turbulence)
  - Inaccuracy due to simplifications(incompressible, steady)

“ Solving the right equations”
(Roache 1998)

CFD (Computational Fluid Dynamics)





Or

Colourful
Fluid
Dynamics

Reference

Roland, Stull “Introduction to boundary layer”.

Launder, B. E., Reece, G. J. and Rodi, W. (1975), "Progress in the Development of a Reynolds-Stress Turbulent Closure.", Journal of Fluid Mechanics, Vol. 68(3), pp. 537-566.

Van Dyke, M. (1982) “An Album of Fluid Motion”.

Spalding, D. B.(1972) “ A Novel Finite Difference Formulation for Differential Expressions Involving both First and Second Derivatives, Int. F. Numer. Methodes Eng., Vol. 4 pp. 551.

Ferziqer, J. H., Peric, M. (2002) “ Computational Methods for Fluid Dynamics”.

Versteeg, H. K., Malalasekara, W. “An introduction to Computational Fluid Dynamics The Finite Volume Method”.

Best practice guidelines sample:

AIAA guide (1998)

ERCOFTAC guidelines (2000)

COST 732 Best Practice Guideline for the CFD simulation of flows in the urban environment (2007)

Roache PJ. “Quantification of uncertainty in computational fluid dynamics.” Annual Reviews in Fluid Mechanics, 29, 123-160