FAKULTA STROJNÍ TUL

Introduction to Computational Fluid Dynamics (CFD)

Lecture 1, 2 (Part 2)

jan.kracik@tul.cz

Agenda – Lecture 1, 2

- Approaches to solving engineering problems
- CFD (What? Why? How?)
- Fundamental equations
- Numerical procedure
- Some CFD applications
- Turbulence in CFD

Investigation approaches

Experimental investigation

- ❑ Most reliable information
- ❑ Can use both full-scale and small-scale tests
- ❑ Not free from errors

Theoretical calculation

- ❑ Usually set of differential equations
- ❑ Solution exists only for a narrow range of practical problems

Numerical calculation (CFD)

- ❑ Finite number of domain elements (discretization)
- ❑ Set of algebraic equations
- ❑ Solution exists almost for each practical applications

Advantages/Disadvantages of theoretical approach

Advantages

- Speed
- Low cost
- Information completeness
- Ability to simulate both ideal and real conditions

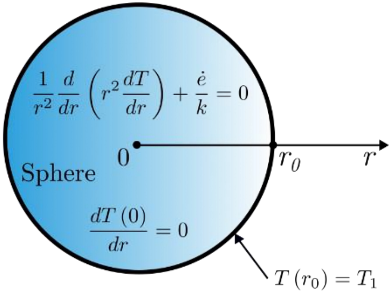
Disadvantages

- Mathematical model
- Problem complexities

Remember: Experiment leads, computation follows.

Why numerical methods?

Heat conduction in a sphere

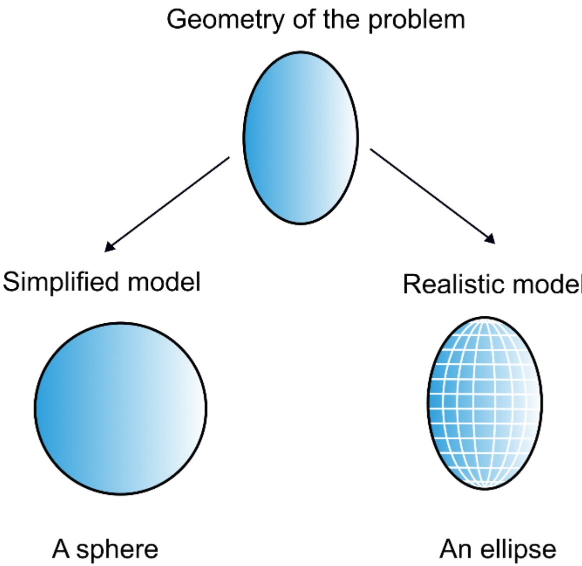


Governing differential equation:

$$\frac{1}{r^2}\frac{d}{dr}\left(r^2\frac{dT}{dr}\right)+\frac{\dot{e}}{k}=0$$

Solution:

$$T(r)=T_1+\frac{\dot{e}}{6k}\left(r_0^2-r^2\right)$$



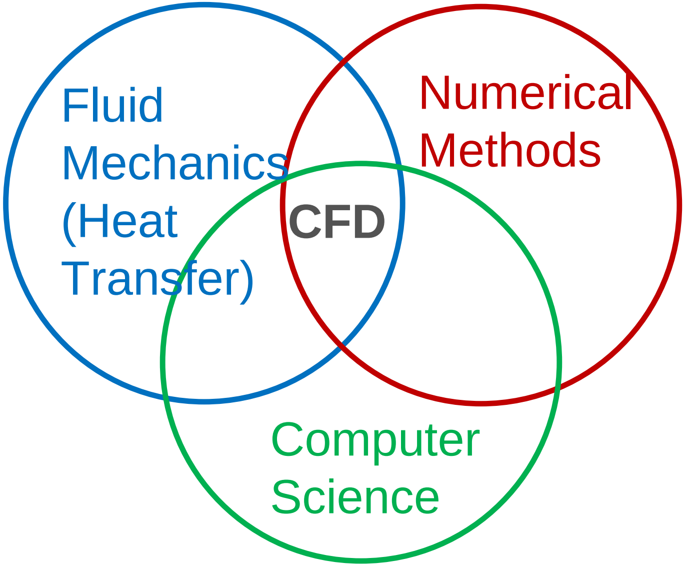
Exact (analytical) solution of model, but crude solution of actual problem

Approximate (numerical) solution of model, but accurate solution of actual problem

Why prefer numerical approach to analytical?

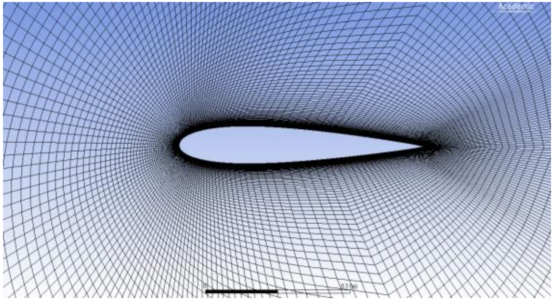
- **Limitations** (geometry, variable HTC, temperature dependent k, …)
- **Better modelling** (“approximate” solution of a realistic model is usually more accurate than the “exact” solution of a crude mathematical model)
- **Flexibility** (parametric studies to answer some “what-if ” questions)
- **Complications** (even when analytical solutions are available, they can be intimidating)
- **Human nature** (ready availability of high-powered computers with sophisticated software packages)

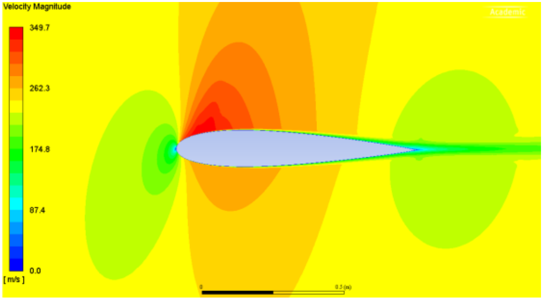
What is CFD?


Fluid Mechanics (Heat Transfer)
Numerical Methods
CFD
Computer Science

How Does CFD Work?

- Numerical analysis and computers come into play.
- Differential equations → Algebraic equations → solution
- The application of CFD to practical problems is often limited by the computational power available.

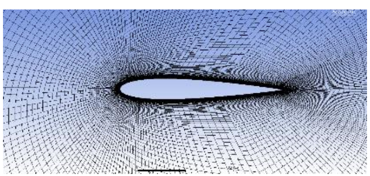


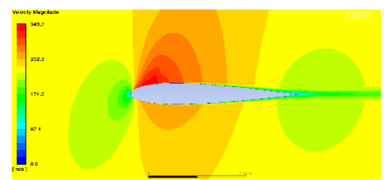


The black box idea

- Valid especially for commercial packages (Ansys Fluent, CFX, Star CCM+).
- More sophisticated codes are also available (OpenFOAM).
- User inputs (geometry, mesh, boundary conditions, material properties, solver settings).
- Turn the crank, and get the results (color pictures).
- By a thorough understanding the black box, we can use the tool effectively.

User inputs → Black box → Color pictures and other results





Set of Fundamental Equations

1) Conservation of mass

$$\boxed{\frac{\partial \rho}{\partial t} + \nabla \cdot [\rho \mathbf{U}] = 0} \qquad (1)$$

2) Conservation of momentum

$$\boxed{\frac{\partial [\rho \mathbf{U}]}{\partial t} + \nabla \cdot \{\rho \mathbf{U}\mathbf{U}\} = [\mathbf{f_s}] + [\mathbf{f_V}]} \qquad (2)$$

3) Conservation of energy

$$\boxed{\frac{\partial (\rho e_{total})}{\partial t} + \nabla \cdot [\rho e_{total} \mathbf{U}] = \mathbf{f_s} \cdot \mathbf{U} + \mathbf{f_V} \cdot \mathbf{U} - \nabla \cdot [\mathbf{q}]} \qquad (3)$$

Note:

$$e_{total} = e_{internal} + \frac{1}{2} \mathbf{U} \cdot \mathbf{U} \quad (4) \qquad\qquad \mathbf{q} = -k\nabla T \quad (5)$$

Set of Fundamental Equations (2)

1) Conservation of mass

$$\frac{\partial \rho}{\partial t} + \nabla \cdot [\rho \mathbf{U}] = 0 \qquad (6)$$

2) Conservation of momentum

$$\frac{\partial [\rho \mathbf{U}]}{\partial t} + \nabla \cdot \{\rho \mathbf{U}\mathbf{U}\} = [\mathbf{f_s}] + [\mathbf{f_V}] = \nabla \cdot \boldsymbol{\sigma} + [\rho \mathbf{g}] = -\nabla p + \nabla \cdot \boldsymbol{\tau} + [\rho \mathbf{g}] \qquad (7)$$

3) Conservation of energy

$$\frac{\partial (\rho e_{total})}{\partial t} + \nabla \cdot [\rho e_{total} \mathbf{U}] = \mathbf{f_s} \cdot \mathbf{U} + \mathbf{f_V} \cdot \mathbf{U} - \nabla \cdot [\mathbf{q}] =$$
$$= -\nabla \cdot [p\mathbf{U}] + \nabla \cdot [\boldsymbol{\tau} \cdot \mathbf{U}] + [\rho \mathbf{g}] \cdot \mathbf{U} + \nabla \cdot [k \nabla T] \qquad (8)$$

Note:

$$e_{total} = e_{internal} + \frac{1}{2} \mathbf{U} \cdot \mathbf{U} \qquad (9) \qquad\qquad \mathbf{q} = -k\nabla T \qquad (10)$$

Numerical Simulation Procedure

start

1) Mathematical model
2) Physical model (steady, incompressible, etc.)
3) Domain discretization → Numerical mesh
4) Equation discretization → A set of algebraic eqs.
5) Solution method and boundary conditions settings
6) Convergence criteria

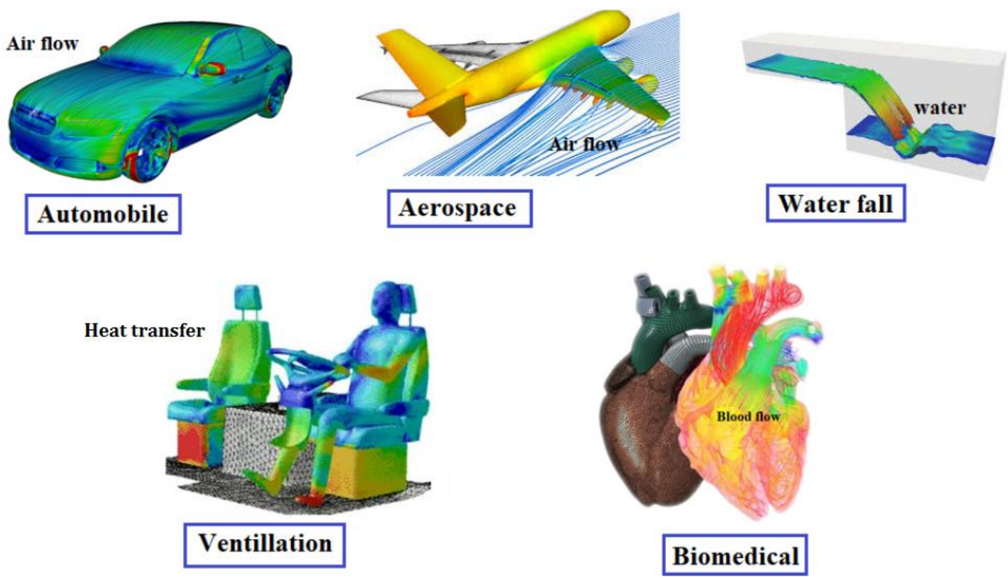
} Pre-processing (1–6)

7) Solution calculation } Processing or solving

finish

8) Results interpretation and presentation } Post-processing

Applications of CFD

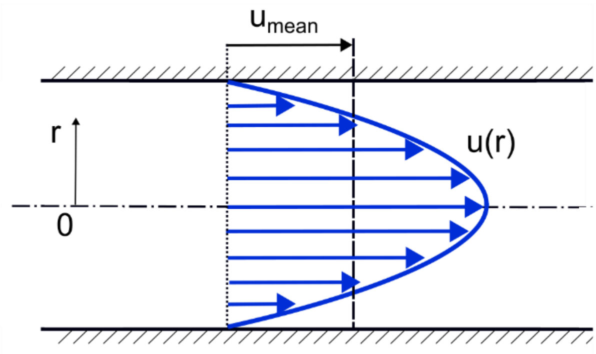


Source: https://cfdflowengineering.com/scope-of-cfd-modeling-career-and-job-opportunities/

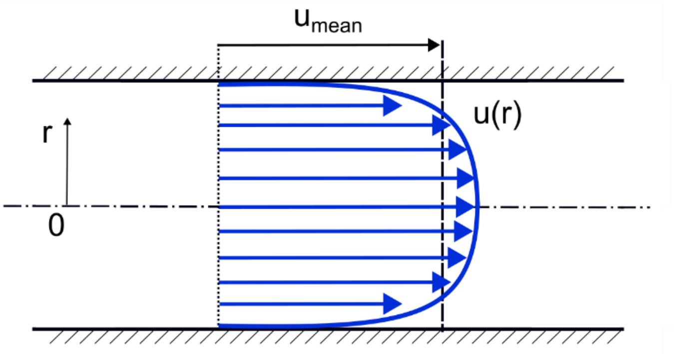
Solving of turbulence in CFD

- Flow can be laminar, turbulent (more frequent), or transitional (complex to solve).
- Most flows in practice are turbulent.
- Laminar solutions are almost exact (Mesh resolution, BCs).
- Resolution of complex turbulent flows is challenging and not feasible these days.

Laminar profile



Turbulent profile

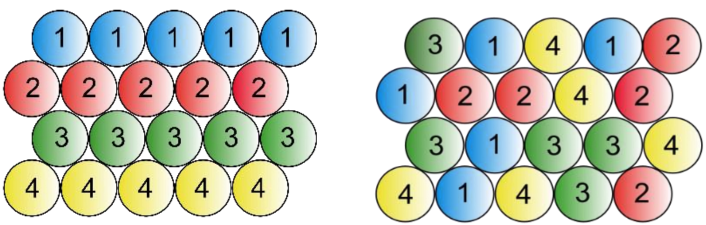


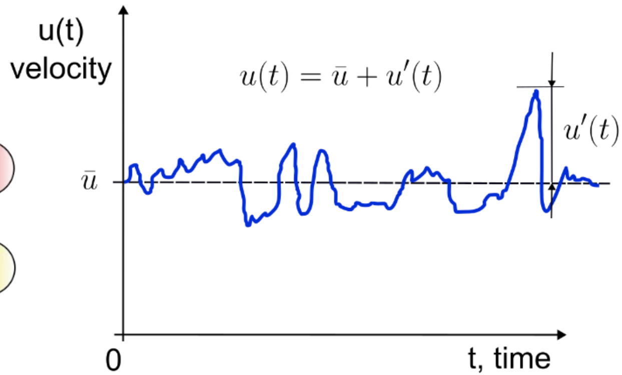
Solving of turbulence in CFD (2)

- Direct Numerical Simulation (DNS) is not useful for practical engineering problems.
- It would require a very fine mesh to capture all turbulent motions.
- Therefore, we must rely on experiments and empirical correlations.

Before turbulence

After turbulence





Solving of turbulence in CFD (3)

- Turbulent flows are characterized by random and rapid fluctuations of swirling regions (= eddies).
- We need to capture these turbulent structures somehow.
- One of the options how to do it is to get inspiration in molecular motion fluctuations.
- In laminar flows, the molecular viscosity causes the shear stress.
- In turbulent flows, this shear stress is still present and additional stresses arise from the turbulent fluctuations.

$$\boxed{\boldsymbol{\tau}_{\text{total}} = \boldsymbol{\tau}_{\text{laminar}} + \boldsymbol{\tau}_{\text{turbulent}}} \tag{11}$$

Solving of turbulence in CFD (4)

- **The laminar component** of the total shear stress can be expressed as:

$$\boldsymbol{\tau}_{\mathrm{laminar}} = -\mu \frac{\partial \bar{U}}{\partial r} = -\mu \frac{\partial \bar{U}}{\partial y} \tag{12}$$

- In an analogous manner, we can express **the turbulent component**:

$$\boldsymbol{\tau}_{\mathrm{turbulent}} = -\mu_t \frac{\partial \bar{U}}{\partial r} = -\mu_t \frac{\partial \bar{U}}{\partial y} \tag{13}$$

- The problem here is that we do **not have $\boldsymbol{\mu_t}$**, which is not a material constant as μ, but rather a property of turbulent flow.

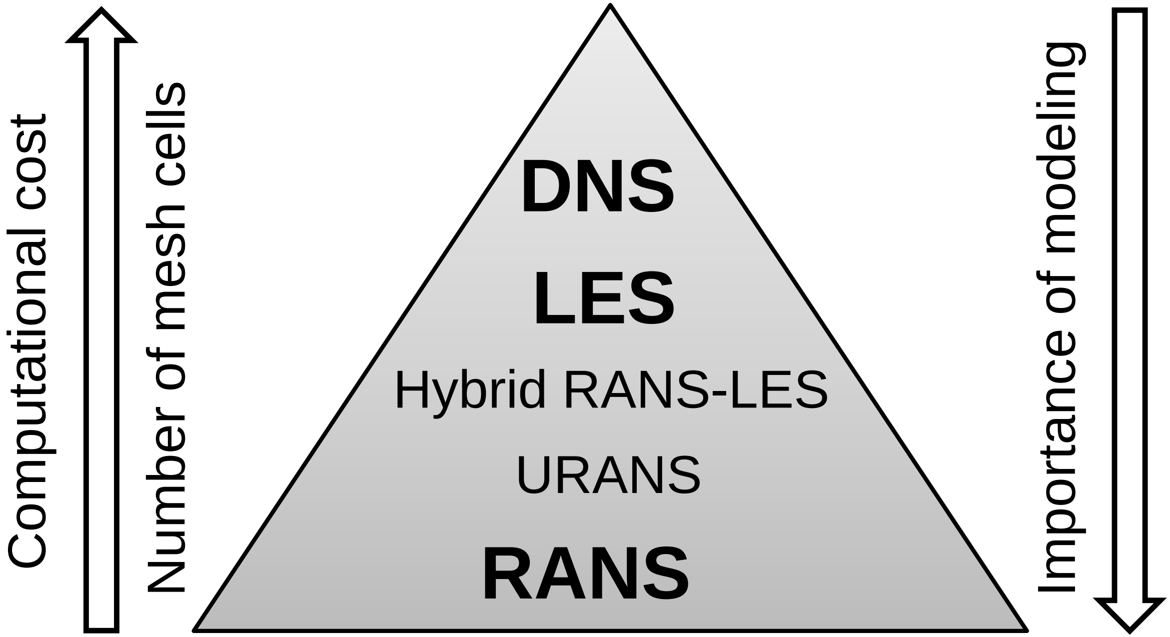
- Note that the turbulent shear stress is often also expressed as:

$$\boxed{\boldsymbol{\tau}_{\mathrm{turbulent}} = -\rho \overline{u'v'}} \tag{14}$$

Solving of turbulence in CFD (5)

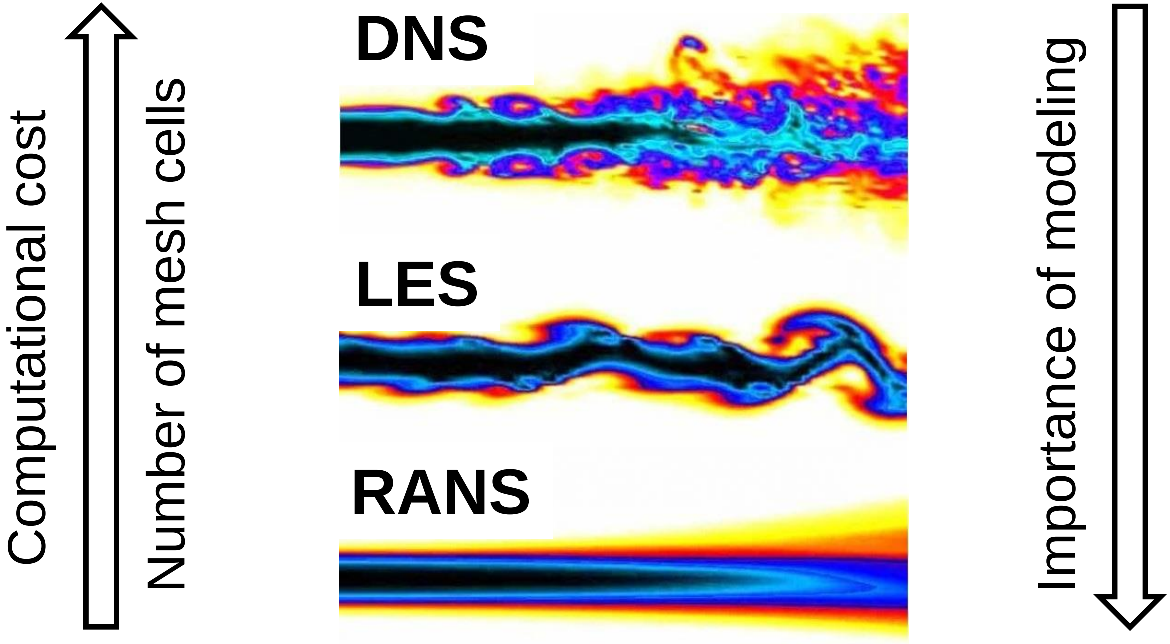
- Most simulations require a model (a coarser mesh can be used).
- No universal model exists for all turbulent flows.
- Turbulence models aim to represent the effect of turbulence via some additional terms or equations.
- Models try to capture the mixing and diffusion caused by turbulent eddies.
- CFD results are only as good as the turbulence model used.

Turbulence models in CFD



DNS - Direct Numerical Simulation
LES - Large Eddy Simulation
RANS - Reynolds-Averaged Navier-Stokes
URANS - Unsteady (transient) RANS

Turbulence models in CFD



DNS - Direct Numerical Simulation
LES - Large Eddy Simulation
RANS - Reynolds-Averaged Navier-Stokes
URANS - Unsteady (transient) RANS

Summary

- CFD can be a handy tool (What? Why? How?)
- Governing equations: **mass**, **momentum**, and **energy**
- Computational process step by step (from pre- to post-processing)
- Some applications
- Treating turbulence phenomena

Thank You!