Introduction to Computational Fluid Dynamics (CFD)

Lecture 1, 2 (Part 2)

jan.kracik@tul.cz

Agenda – Lecture 1

- Approaches to solving engineering problems
- CFD (What? Why? How?)
- Fundamental equations

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

Strengths and weaknesses of theoretical approach

Advantages (Strengths)

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

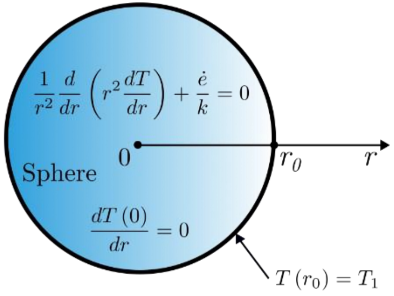
Disadvantages (Weaknesses)

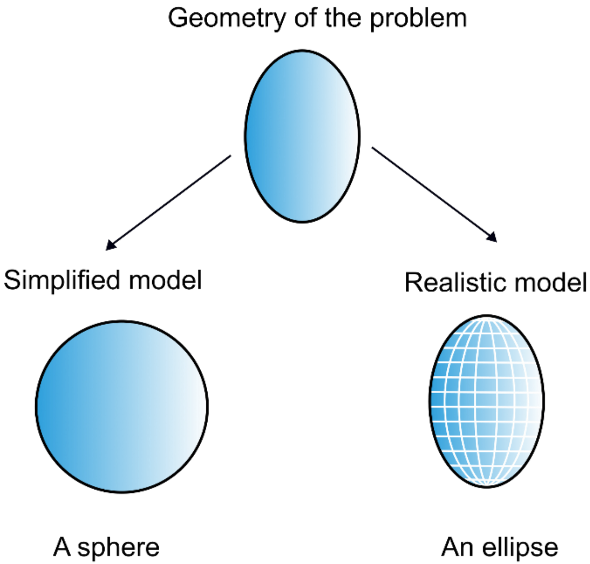
- Mathematical model
- Problem complexities

Remember: Experiment leads, computation follows.

Why use numerical methods?

Problem: 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:

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

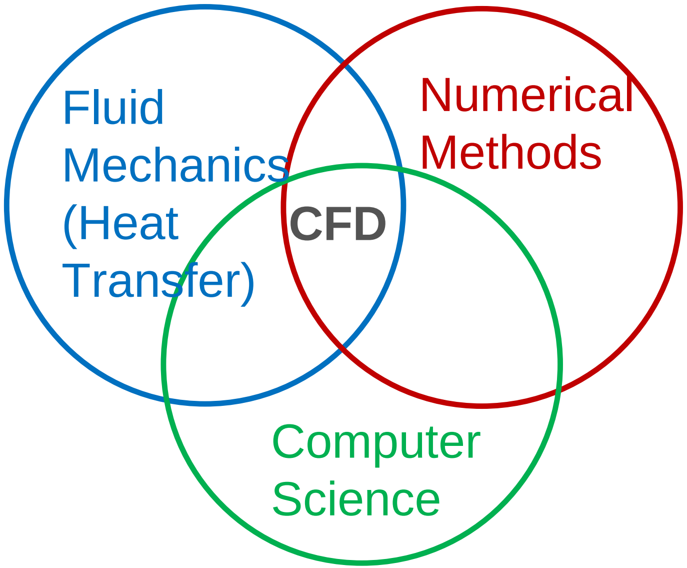
Exact (analytical) solution of model, but crude solution of actual problem

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

Why prefer numerical approach to analytical?

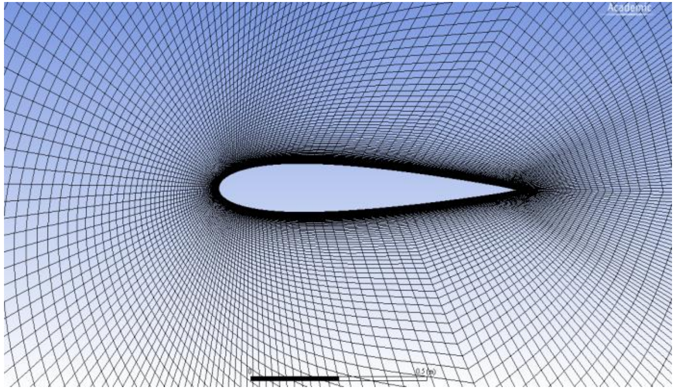
- **Limitations** (geometry, variable HTC, temperature dependent k, …)
- **Better modelling** (“approximate” solution of a realistic model is usually more accurate than the “exact” solution of a crude mathematical model)
- **Flexibility** (parametric studies to answer some “what-if ” questions)
- **Complications**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

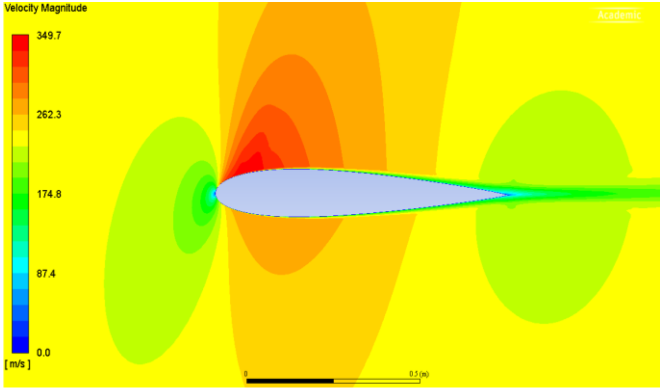
What is CFD?


Fluid Mechanics (Heat Transfer)
Numerical Methods
CFD
Computer Science

What is CFD?

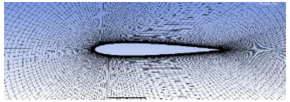
- Numerical analysis and computers come into play.
- Differential equations → Algebraic equations → solution
- The application of CFD to practical problems is often limited by the computational power available.



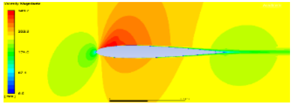


The black box idea

- 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).



User inputs → Black box → Color pictures and other results



Set of Fundamental Equations

1. **Conservation of mass**

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

2. **Conservation of momentum**

$$\frac{\partial [\rho \mathbf{U}]}{\partial t} + \nabla \cdot \{\rho \mathbf{U}\mathbf{U}\} = [\mathbf{f}_s] + [\mathbf{f}_V] \qquad (2)$$

3. **Conservation of energy**

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

Note:

$$e_{total} = e_{internal} + \frac{1}{2} \mathbf{U} \cdot \mathbf{U} \qquad (4)$$

$$\mathbf{q} = -k \nabla T \qquad (5)$$

Set of Fundamental Equations

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**

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

Summary – Lecture 1

- CFD can be a handy tool (What? Why? How?)
- Governing equations: **mass**, **momentum** (Newton‘s 2nd law), and **energy** (1st law of thermodynamics)

Agenda – Lecture 2

- Numerical procedure
- Some CFD applications
- Turbulence in CFD

Numerical Simulation Procedure

Start ↓ Finish

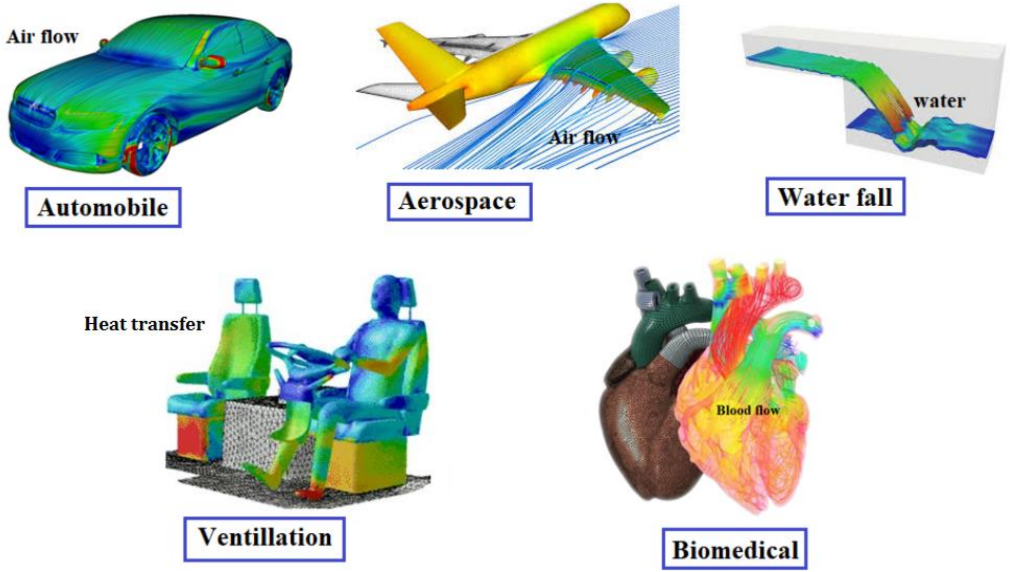
1. Mathematical model
2. Physical model (steady, incompressible, etc.)
3. Domain discretization → Numerical mesh
4. Equation discretization → A set of algebraic eqs.
5. Solution method and boundary conditions settings
6. Convergence criteria
7. Solution calculation
8. Results interpretation and presentation

Steps 1–6: Pre-processing

Step 7: Processing or solving

Step 8: Post-processing

Applications of CFD

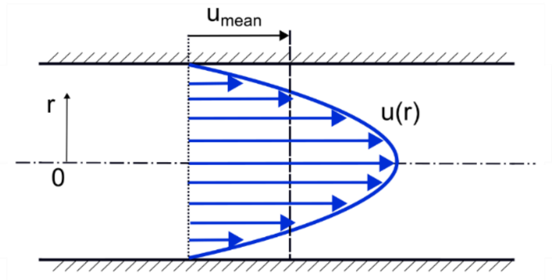


Source: https://cfdflowengineering.com/scope-of-cfd-modeling-career-and-job-opportunities/

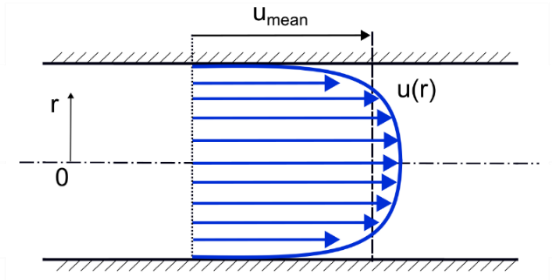
Treatment of turbulence in CFD

- 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



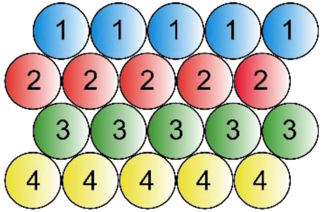
Turbulent profile



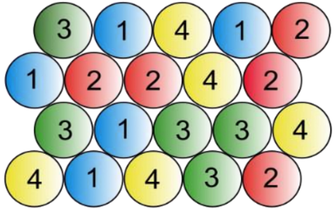
Treatment of turbulence in CFD (2)

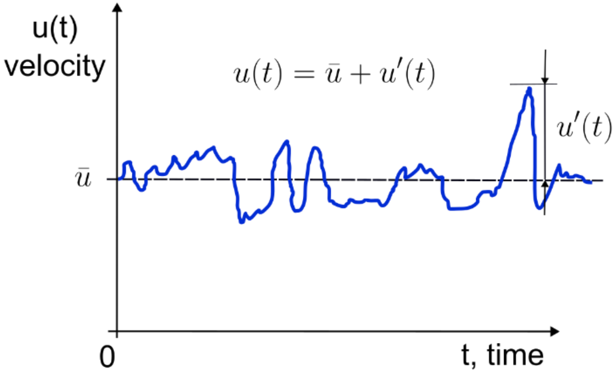
- Direct Numerical Simulation (DNS) is not useful for practical engineering problems.
- It would require a very fine mesh to capture all turbulent motions.
- Therefore, we must rely on experiments and empirical correlations.

Before turbulence



After turbulence





Treatment 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.

$$\boldsymbol{\tau}_{\text{total}} = \boldsymbol{\tau}_{\text{laminar}} + \boldsymbol{\tau}_{\text{turbulent}} \qquad (9)$$

Treatment of turbulence in CFD (4)

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

$$\boldsymbol{\tau}_{\text{laminar}} = -\mu \frac{\partial \bar{U}}{\partial r} = -\mu \frac{\partial \bar{U}}{\partial y} \tag{10}$$

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

$$\boldsymbol{\tau}_{\text{turbulent}} = -\mu_t \frac{\partial \bar{U}}{\partial r} = -\mu_t \frac{\partial \bar{U}}{\partial y} \tag{11}$$

- The problem here is that we do **not have μ_t**, which is not a material constant as μ, but rather a property of turbulent flow.

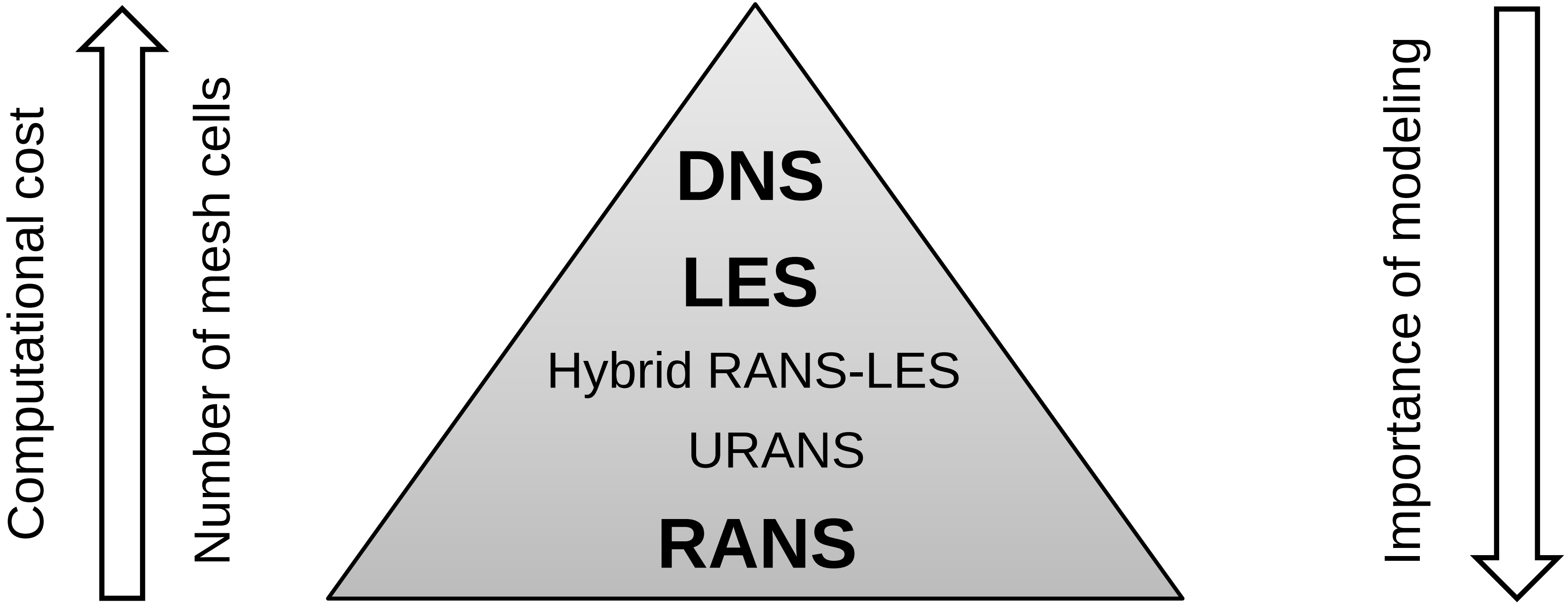
- Note that the turbulent shear stress is often also expressed as:

$$\boldsymbol{\tau}_{\text{turbulent}} = -\rho \overline{u'v'} \tag{12}$$

Treatment of turbulence in CFD (5)

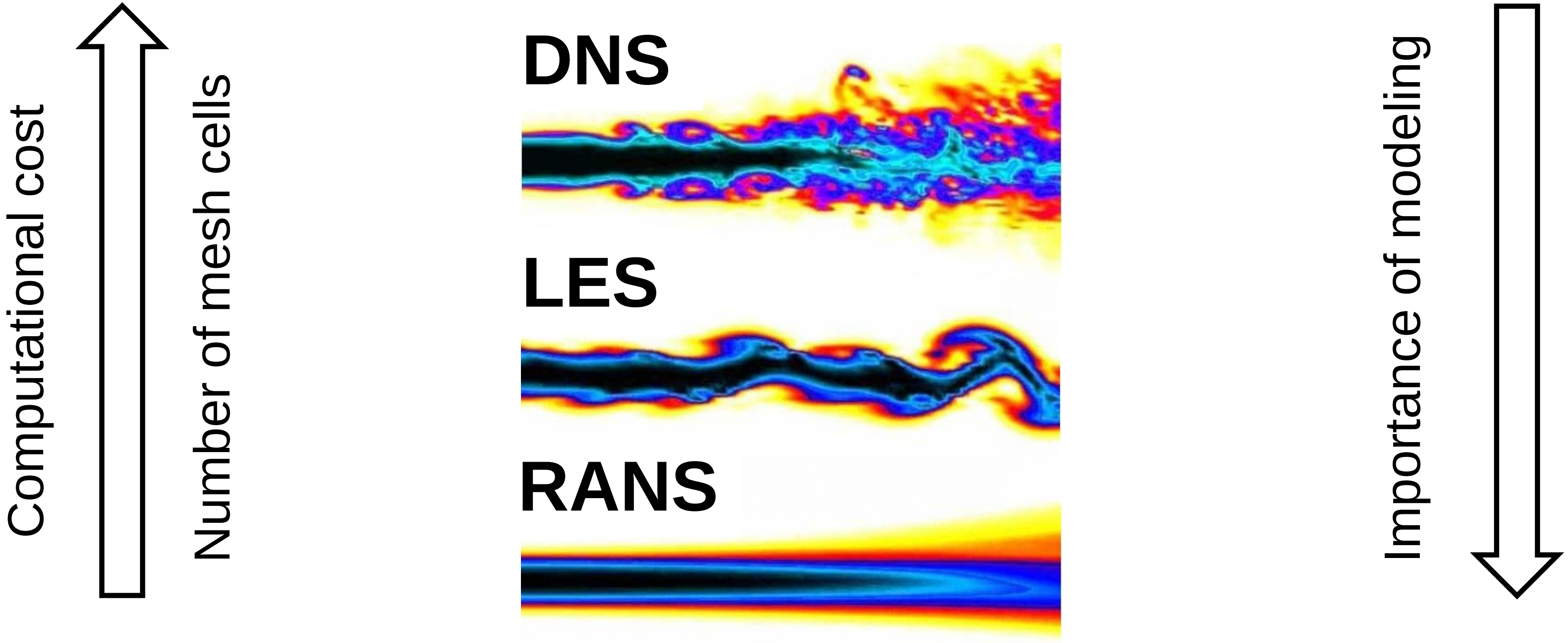
- Most simulations require a model (a coarser mesh can be used).
- No universal model exists for all turbulent flows.
- Turbulence models aim to represent the effect of turbulence via some additional terms or equations.
- Models try to capture the mixing and diffusion caused by turbulent eddies.
- CFD 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 (2)



DNS - Direct Numerical Simulation
LES - Large Eddy Simulation

RANS - Reynolds-Averaged Navier-Stokes
URANS - Unsteady (transient) RANS

Summary – Lecture 2

- Computational process step by step (from pre- to post-processing)
- Some applications
- Treating turbulence phenomena

Thank you.