Introduction

Numerical Flow Simulation

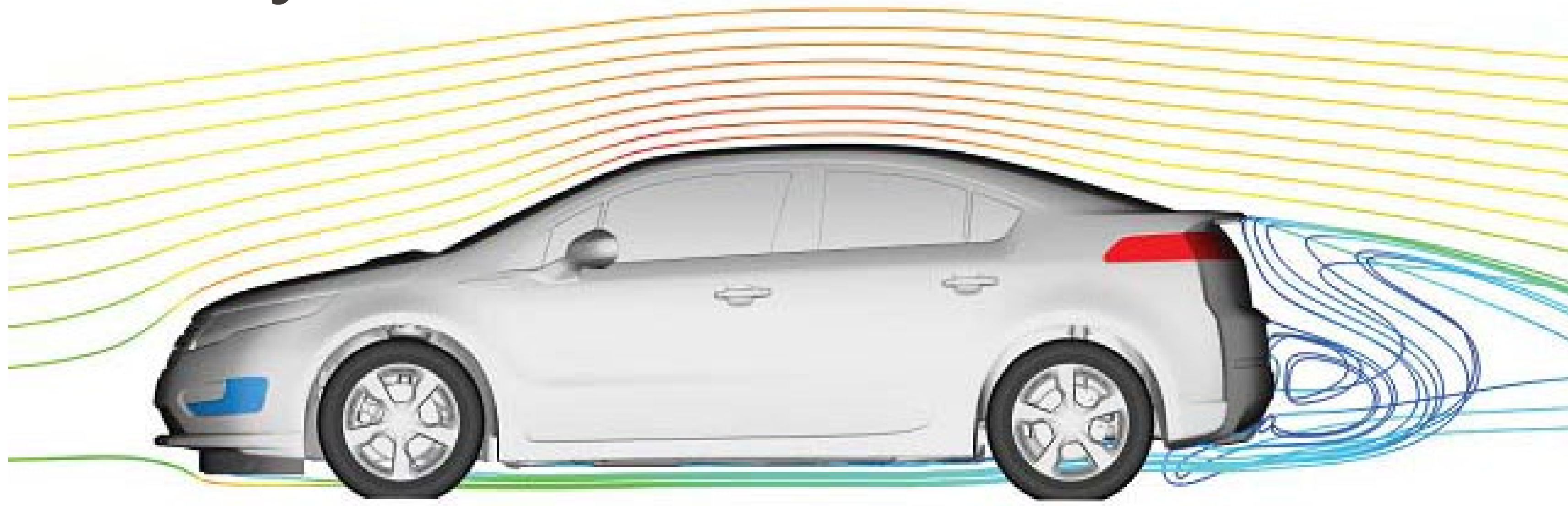
Edouard Boujo

Fall 2025

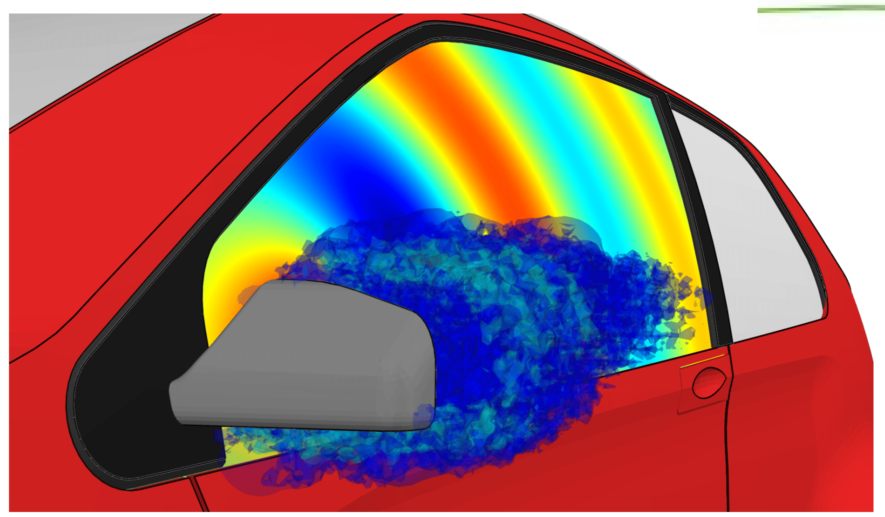
École
polytechnique
fédérale
de Lausanne

Self-introduction

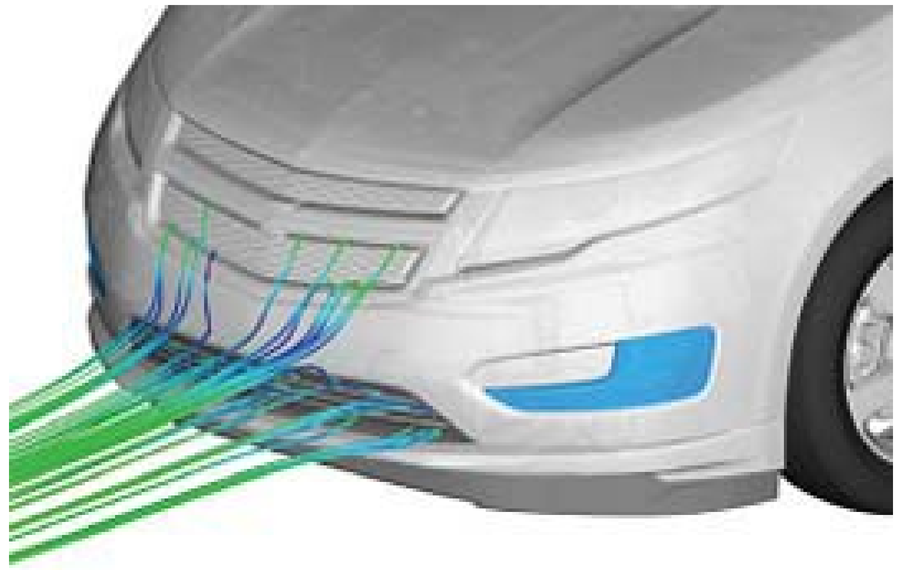
- Previously: CFD engineer in the automotive industry

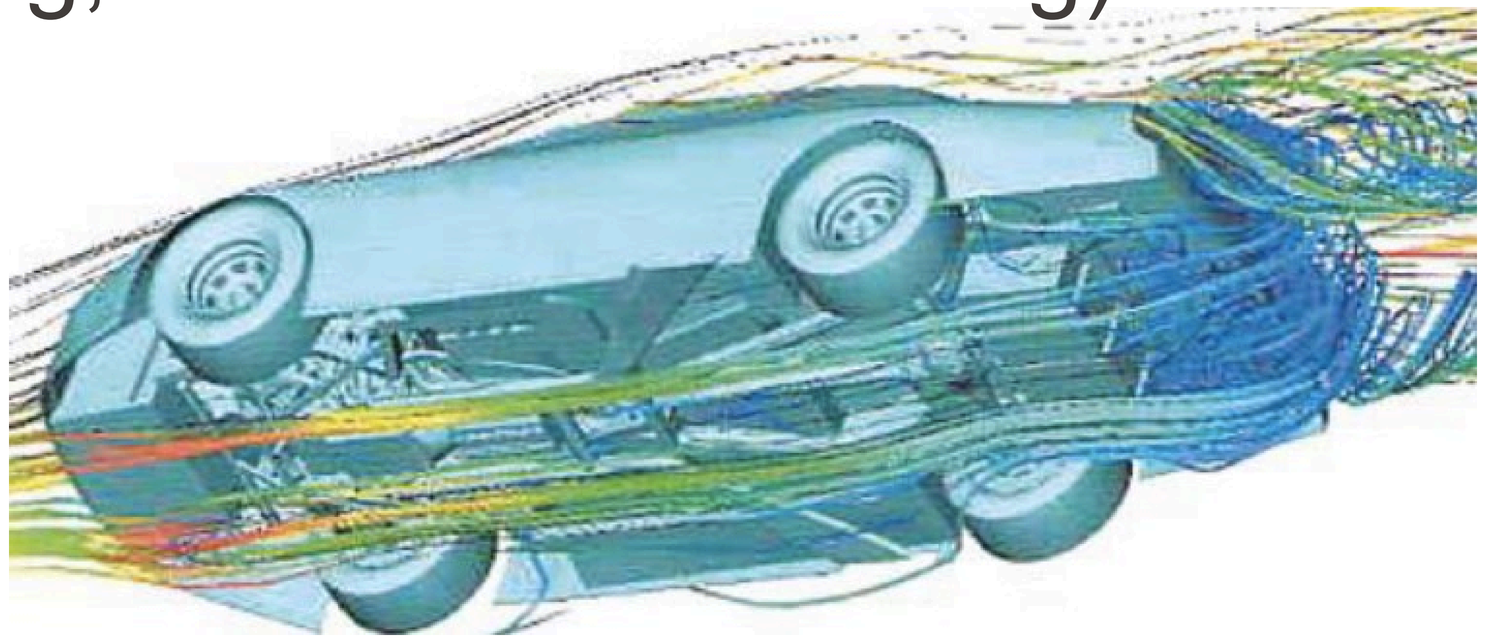


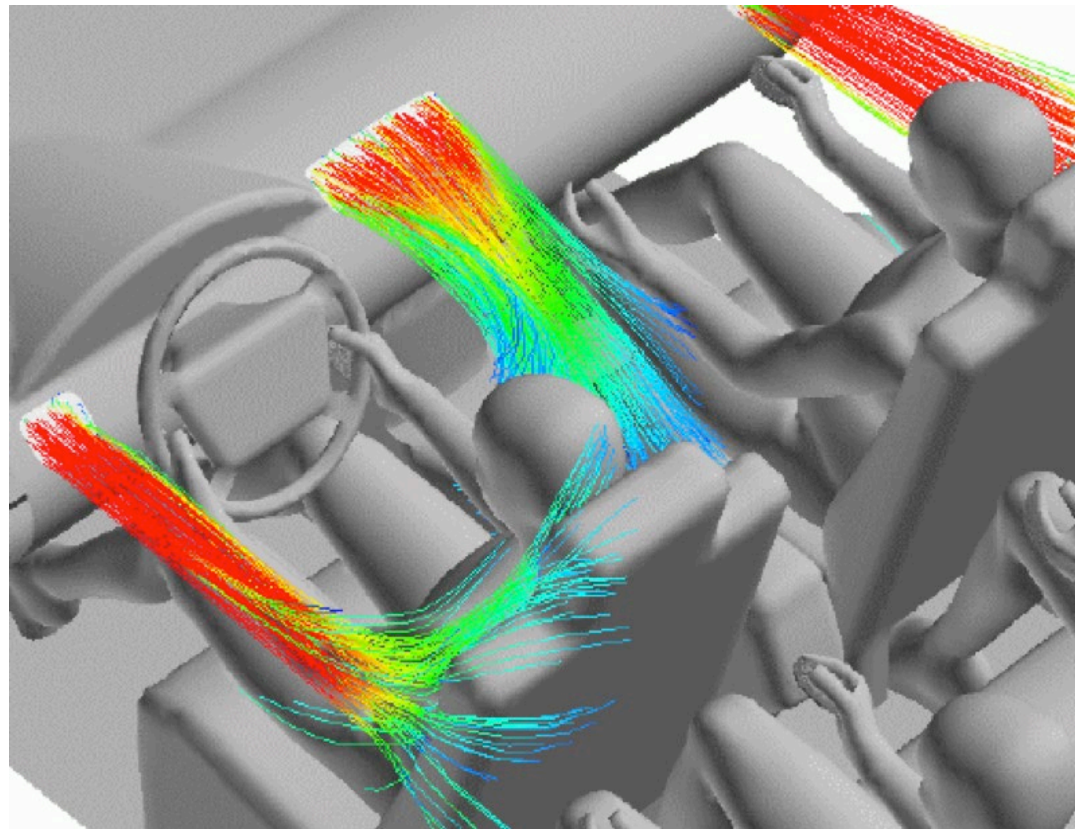
- Method development for
  - Aerodynamics: drag (fuel consumption), lift (handling & stability),



  - Aeroacoustics (“wind noise”),
  - Thermal management (engine cooling, underhood & underbody cooling, cabin air conditioning)

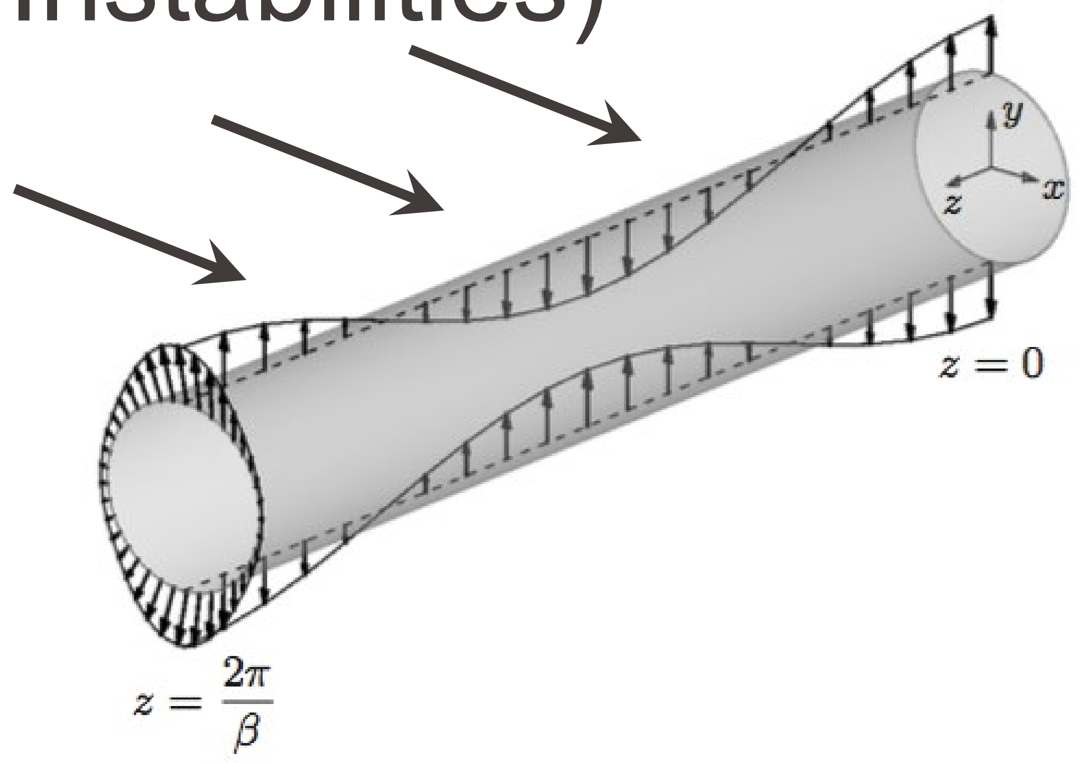




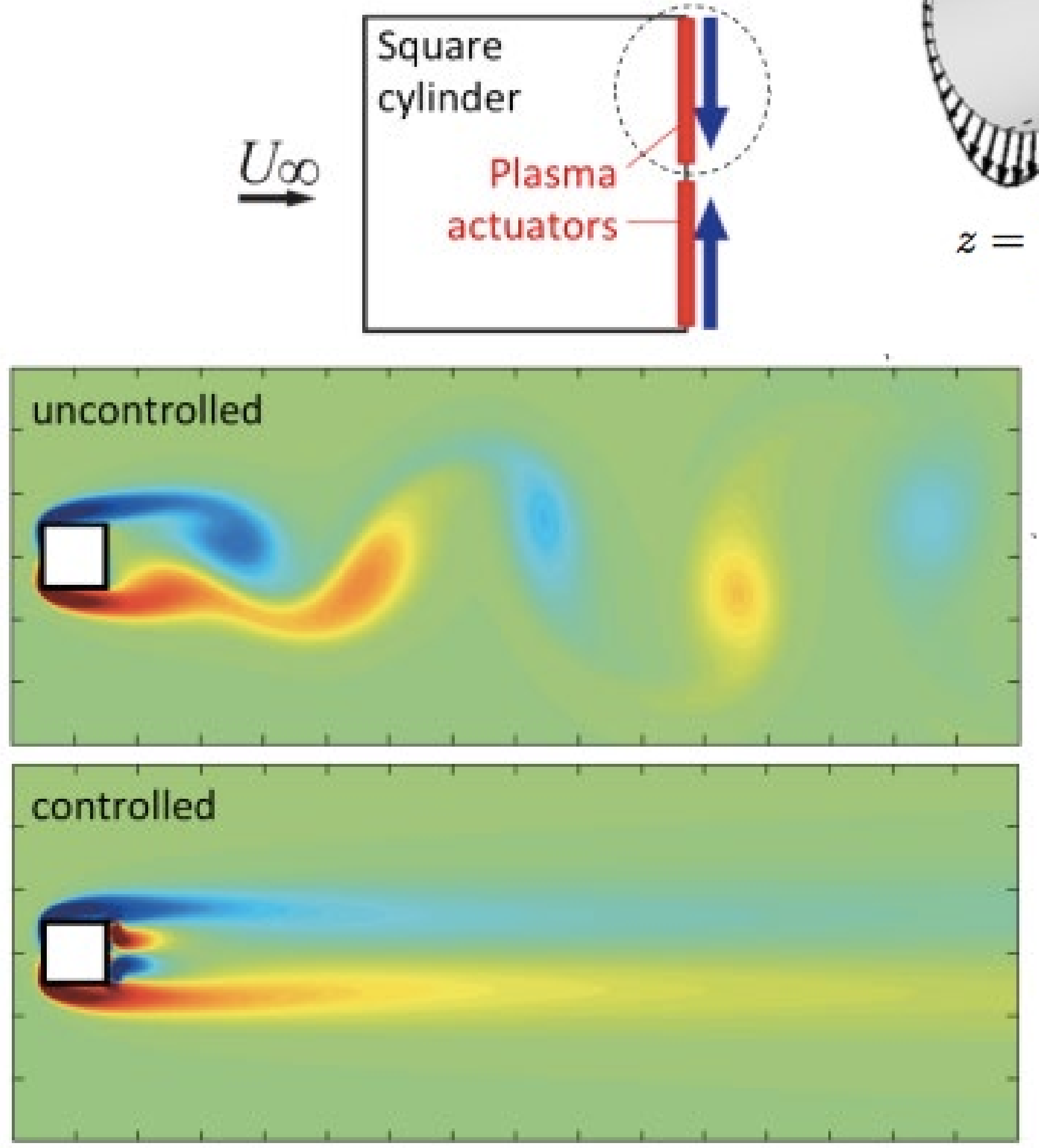


Self-introduction

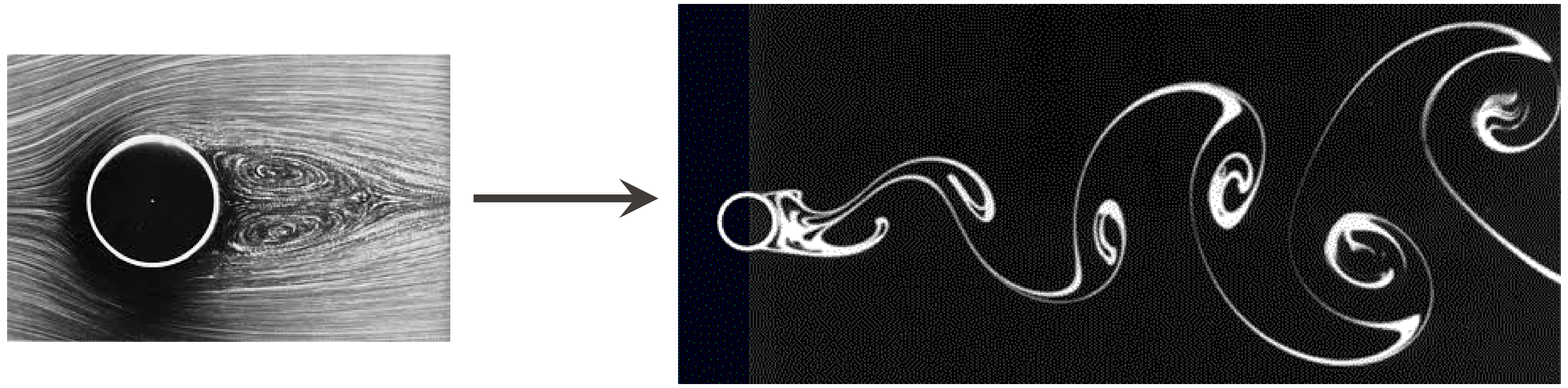
- Currently: scientist at LFMI (Lab. of Fluid Mechanics and Instabilities)

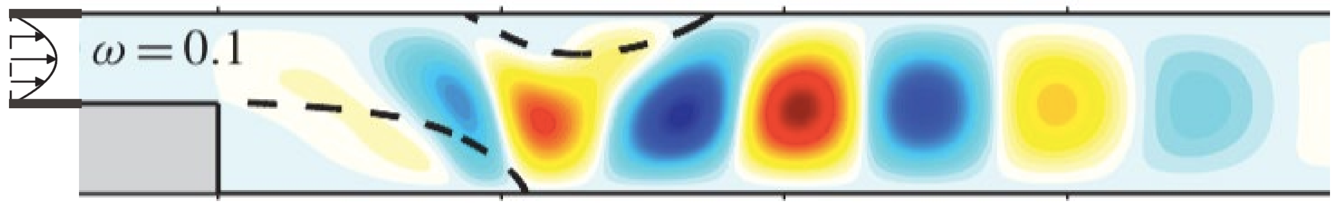


- Research activities:
  - **Flow stability and control**
  - Thermo-/aero-acoustics
  - Fluid-structure interaction
  - Thin-film coating



Numerical Flow Simulation



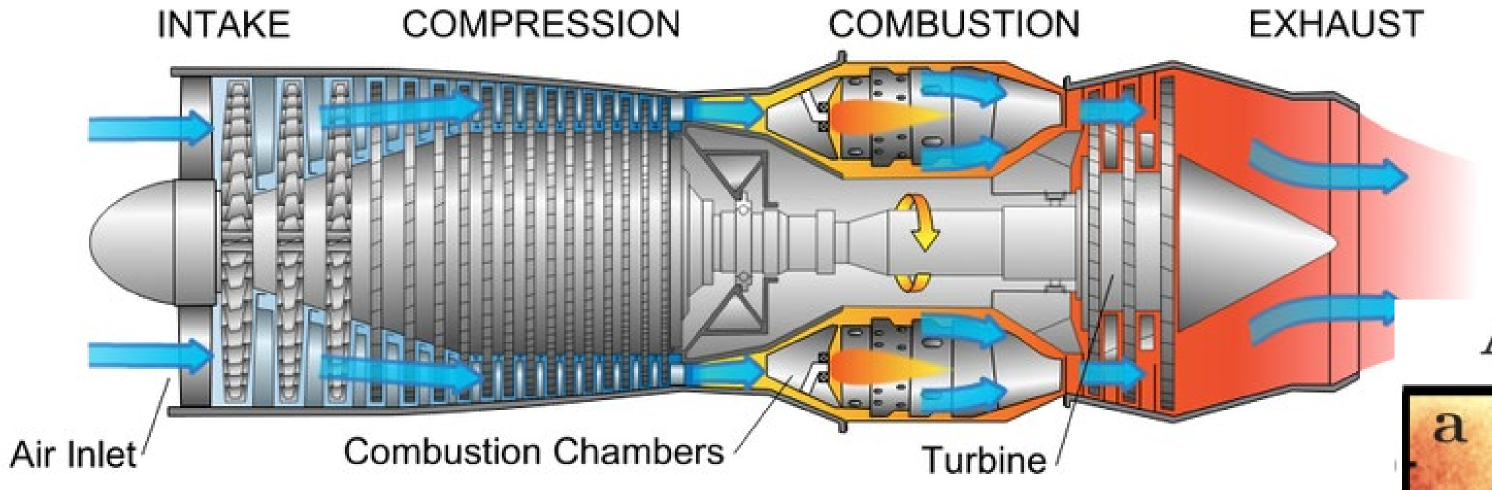


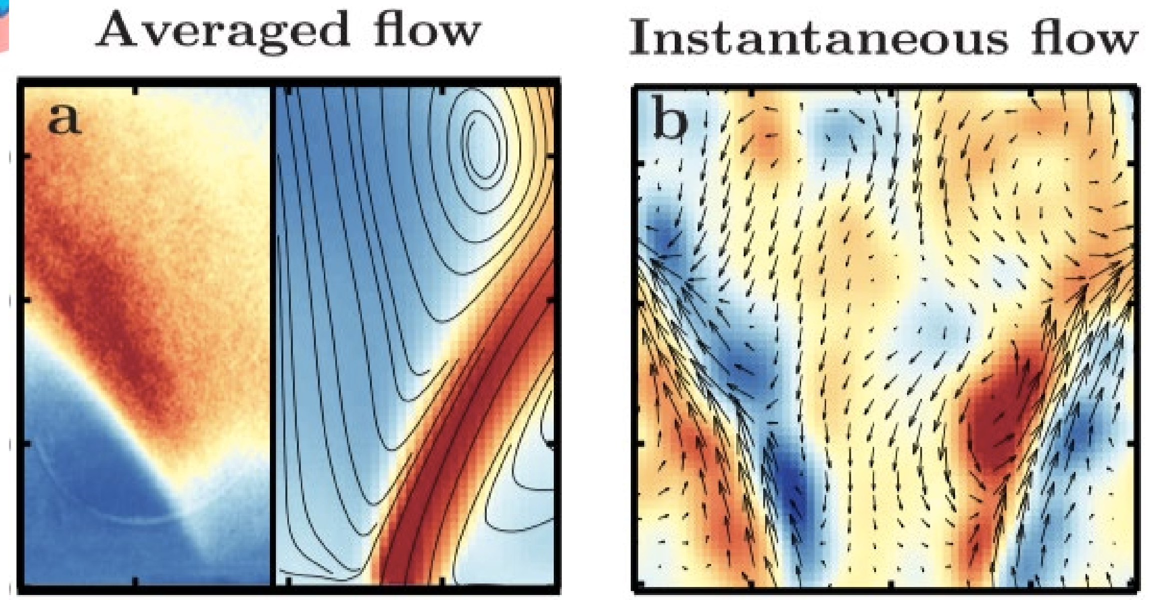
- When/how does a flow becomes unstable? Or strongly amplifies perturbations?
- Systematic methods (optimal control) to improve stability or aerodynamics

Self-introduction

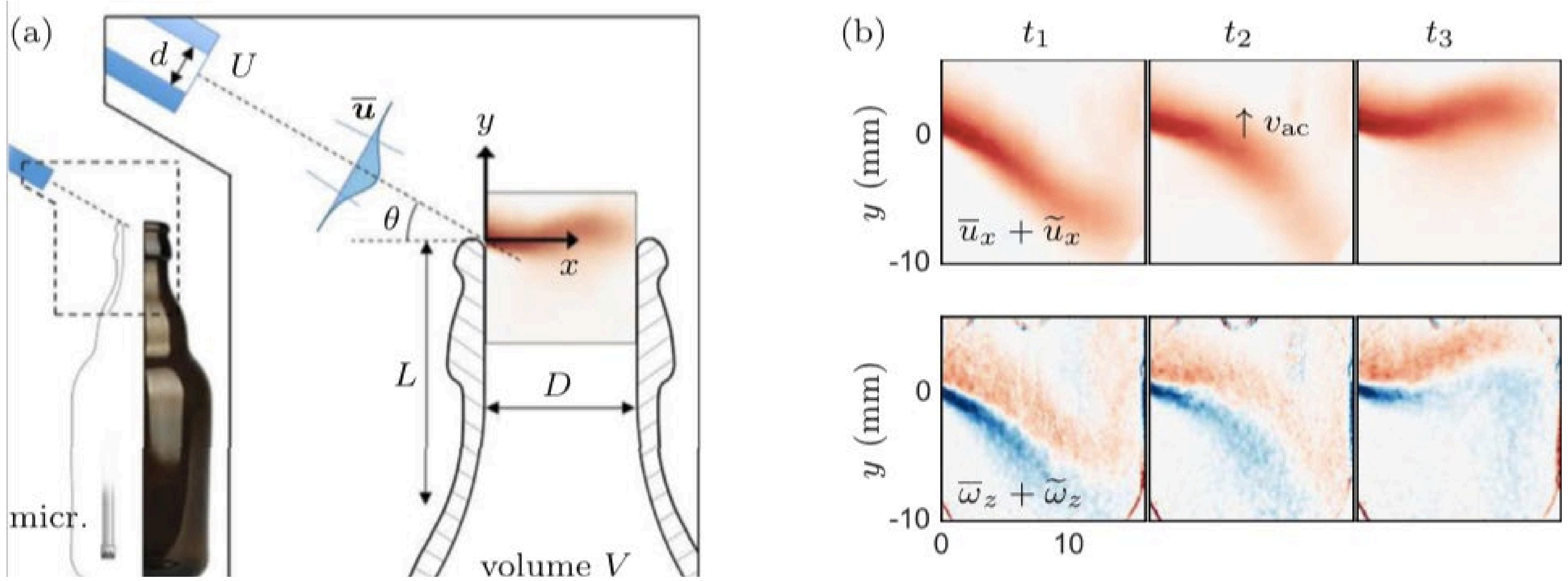
- Currently: scientist at LFMI (Lab. of Fluid Mechanics and Instabilities)

- Research activities:
  - Flow stability and control
  - **Thermo-/aero-acoustics**
  - Fluid-structure interaction
  - Thin-film coating





Numerical Flow Simulation

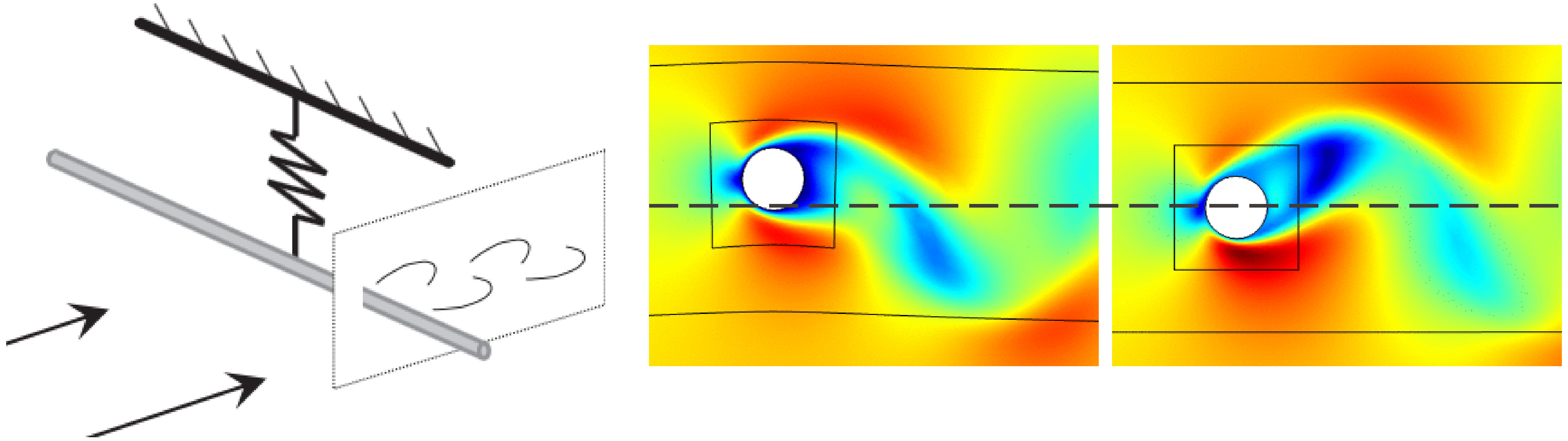


- Coupling between oscillations of pressure and flame, or pressure and jet/mixing layer
- Low-order modelling and identification of instabilities in turbulent systems

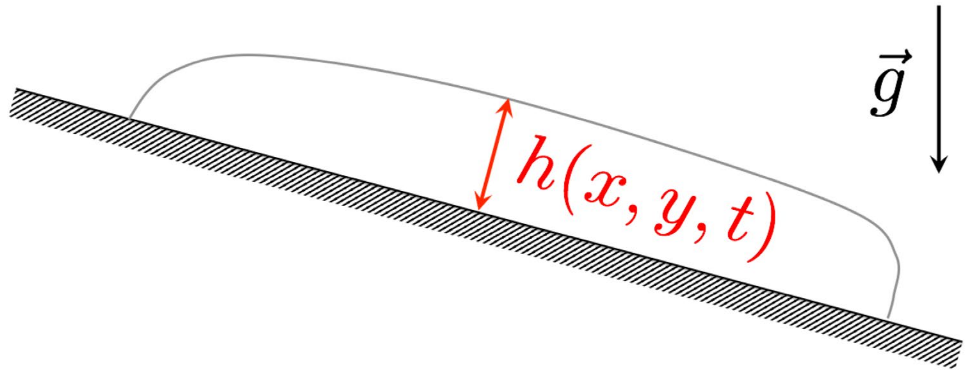
Self-introduction

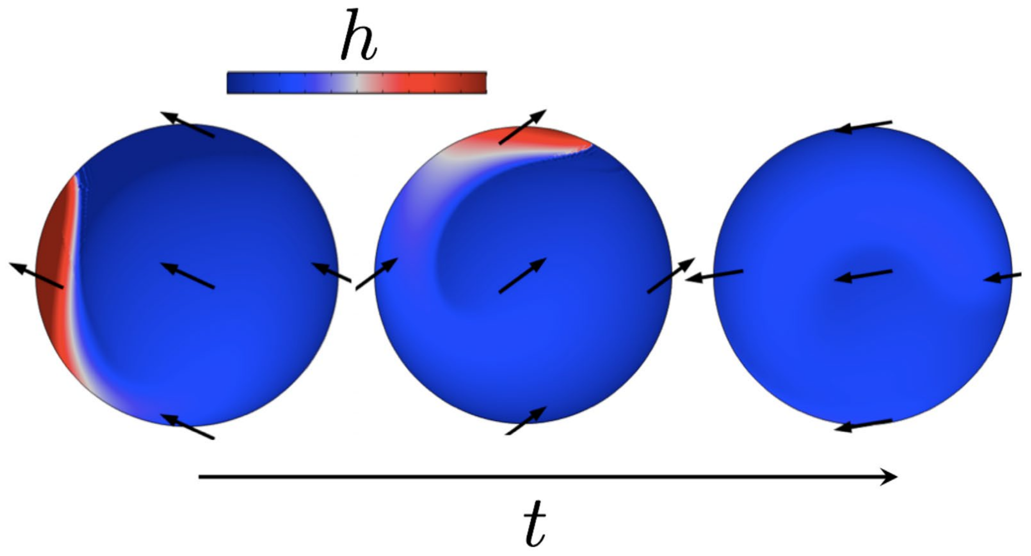
- Currently: scientist at LFMI (Lab. of Fluid Mechanics and Instabilities)

- Research activities:
  - Flow stability and control
  - Thermo-/aero-acoustics
  - **Fluid-structure interaction**
  - **Thin-film coating**



- Energy harvesting, stability properties, low-order modelling





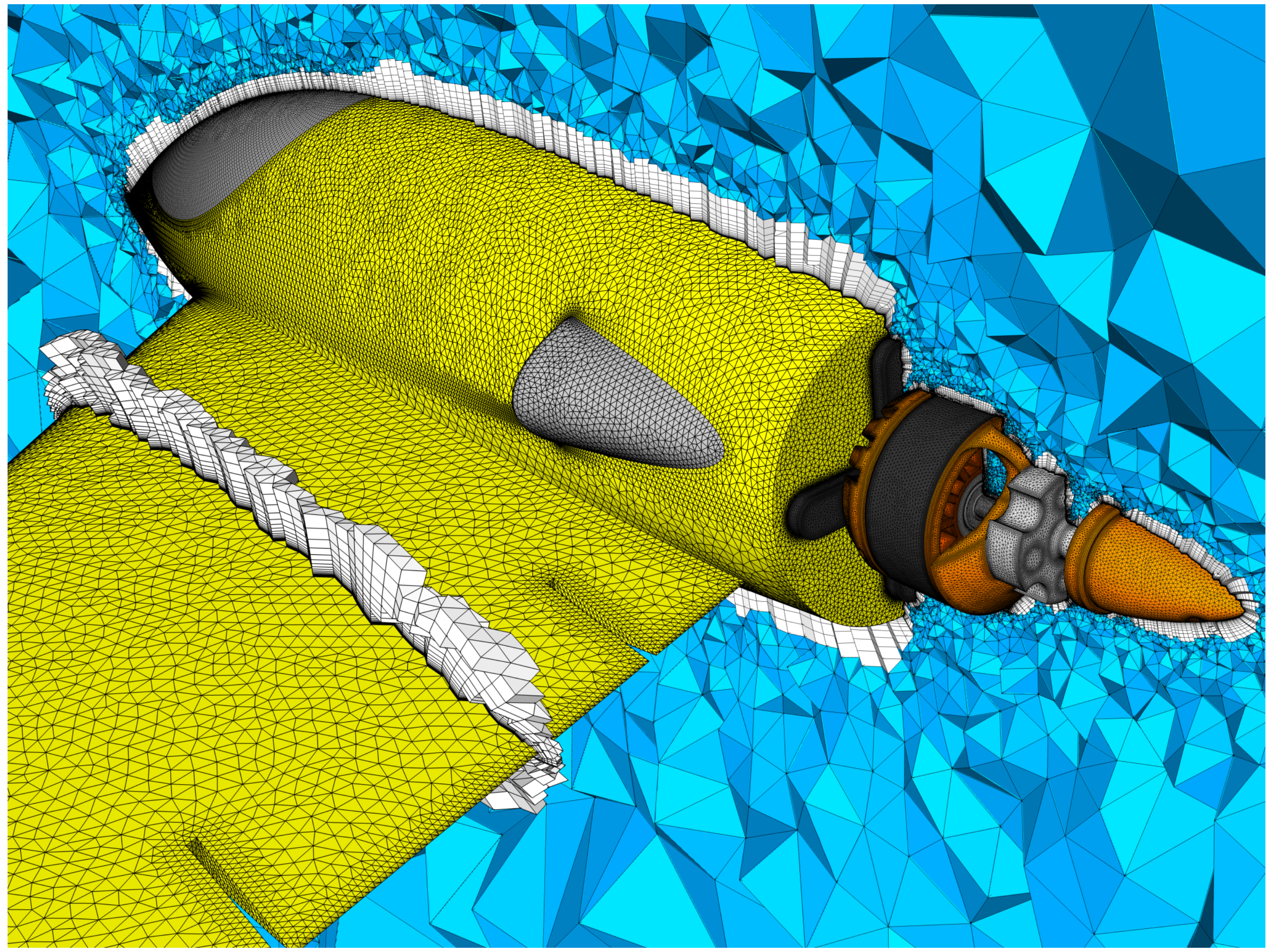
- Optimal motion for uniform coating

Numerical Flow Simulation

Numerical flow simulation

- A.k.a. “Computational Fluid Dynamics” (CFD)

- General idea: discretize fluid domain → transform governing equations (PDEs for continuous variables) into approximate equations (algebraic equations for discrete variables) → solve numerically (computer)



Numerical flow simulation

- Advantages:
  - Can handle complex equations/geometries not solvable analytically
  - Useful when experiments are expensive, difficult, dangerous, time consuming
  - Complete information (time and space-resolved)
  - Parametric studies (+ optimization)

Numerical flow simulation

- Drawbacks/pitfalls:

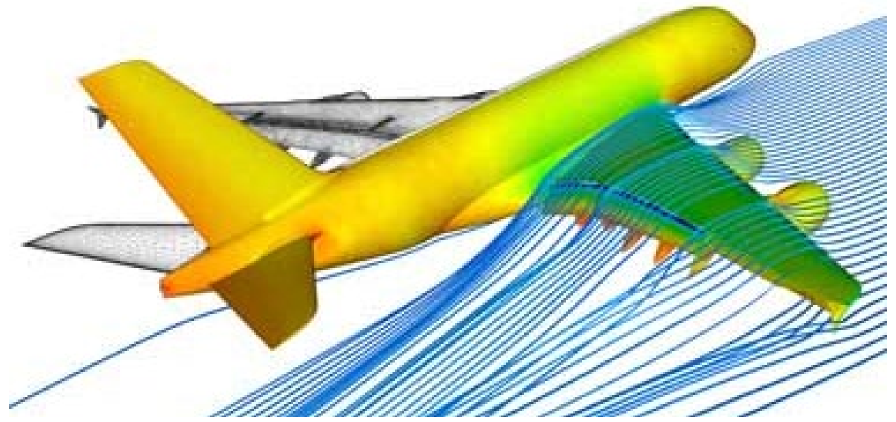
  - Easy to get good-looking but wrong results ("Colourful Fluid Dynamics")
    → requires knowledge (fluid mechanics, numerical methods) AND experience

  - Results only as good as the model: must solve the equations accurately, AND need equations describing the physics correctly (not always available: complex turbulence, non-Newtonian flows, chemical reactions…)

  - No substitute to experiments and theory (but a powerful complementary tool)

Numerical flow simulation

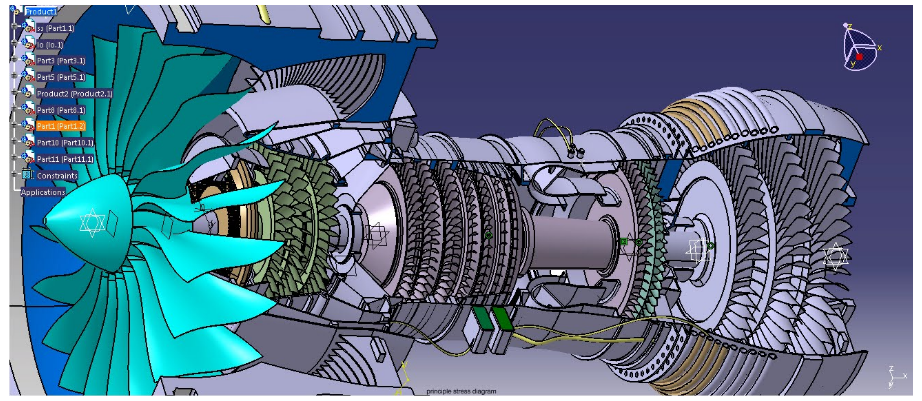
- Broad range of applications in research and engineering:
  - Aerodynamics, aeroacoustics (aircraft, ground vehicles)



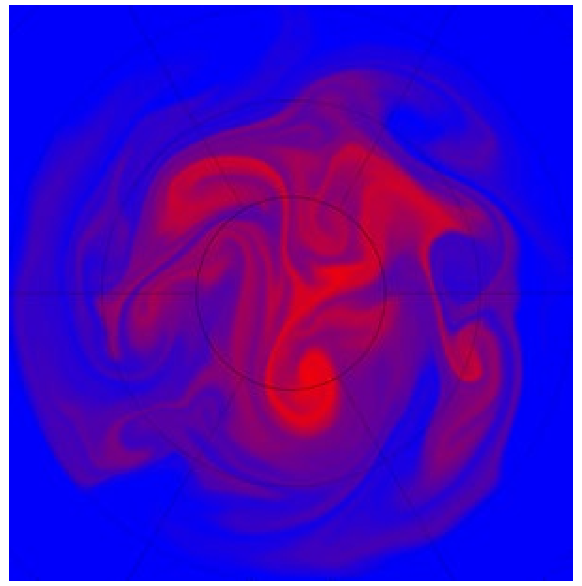
  - Ship hydrodynamics

  - Engines

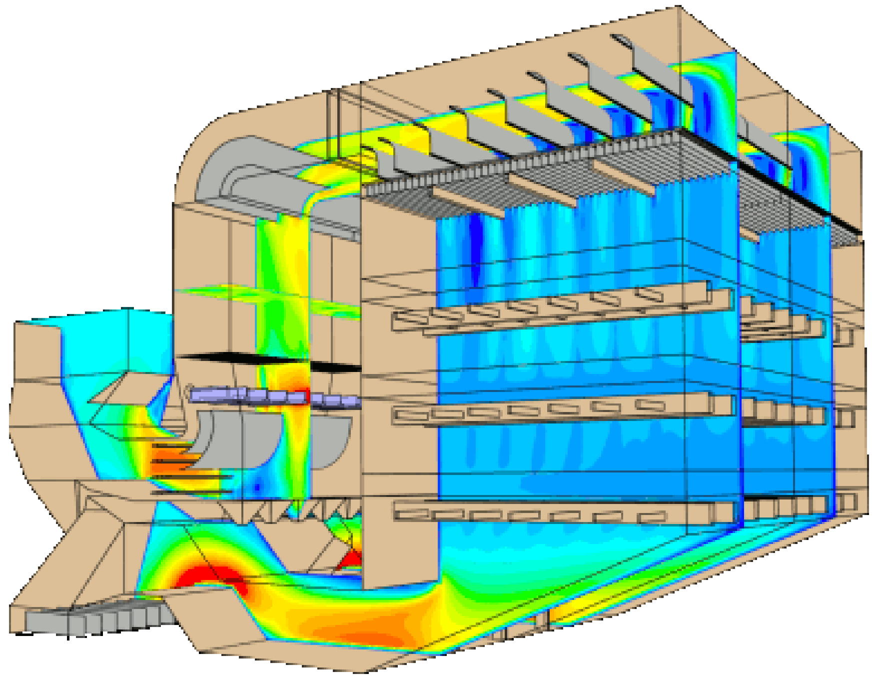
  - Energy: gas turbines, wind turbines, hydro/nuclear power plants



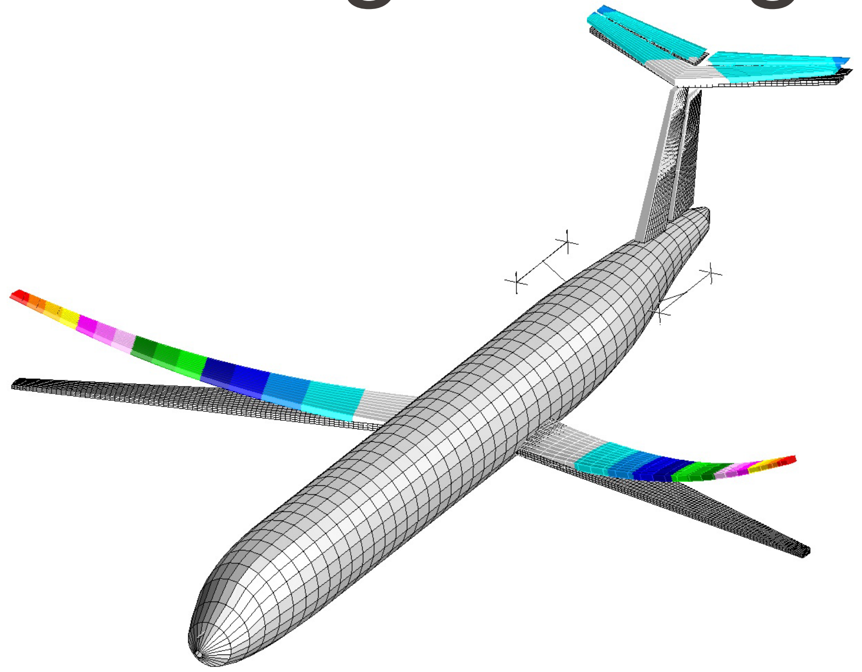
  - Food & chemical processes: foam, emulsions, mixing, particle transport, polymer molding



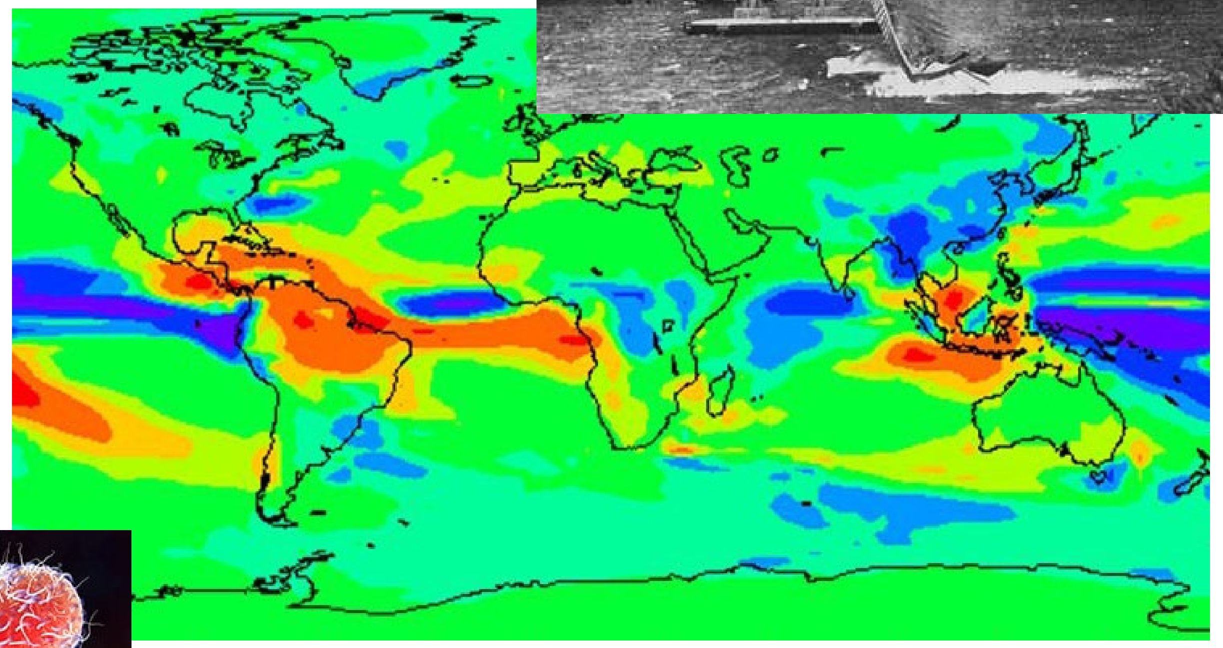
Numerical flow simulation



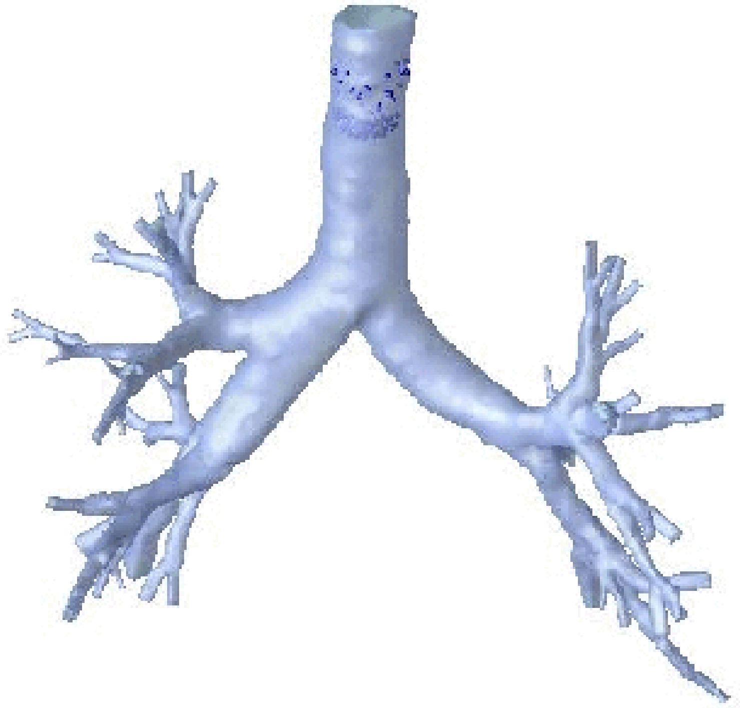
- Broad range of applications in research and engineering:
  - Building thermal management

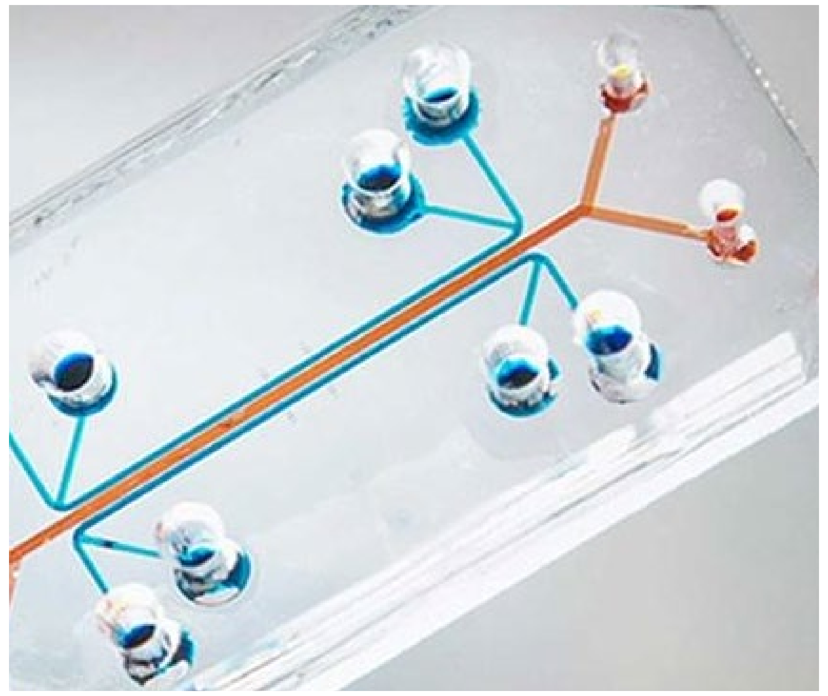


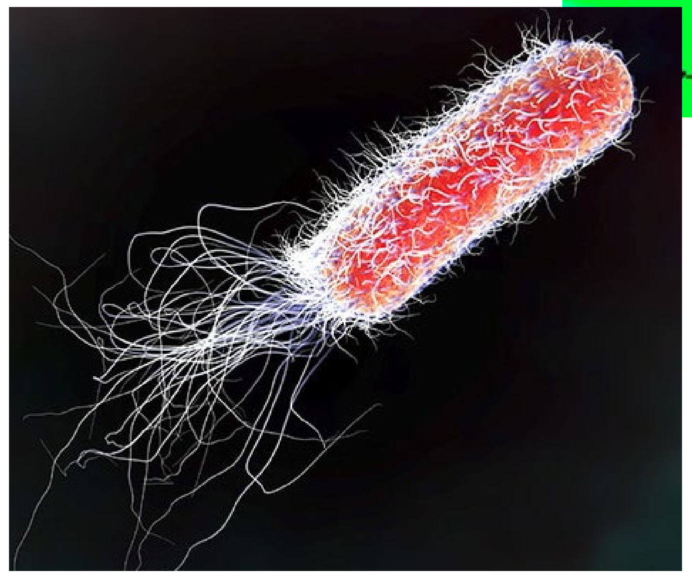
  - Fluid-structure interaction: aeroelasticity, civil engineering, energy harvesting
  - Meteorology: weather prediction, global climate



  - Biomedical: blood flow, lab-on-a-chip devices, microswimmers (bacteria, ciliates, flagellates)
  - …







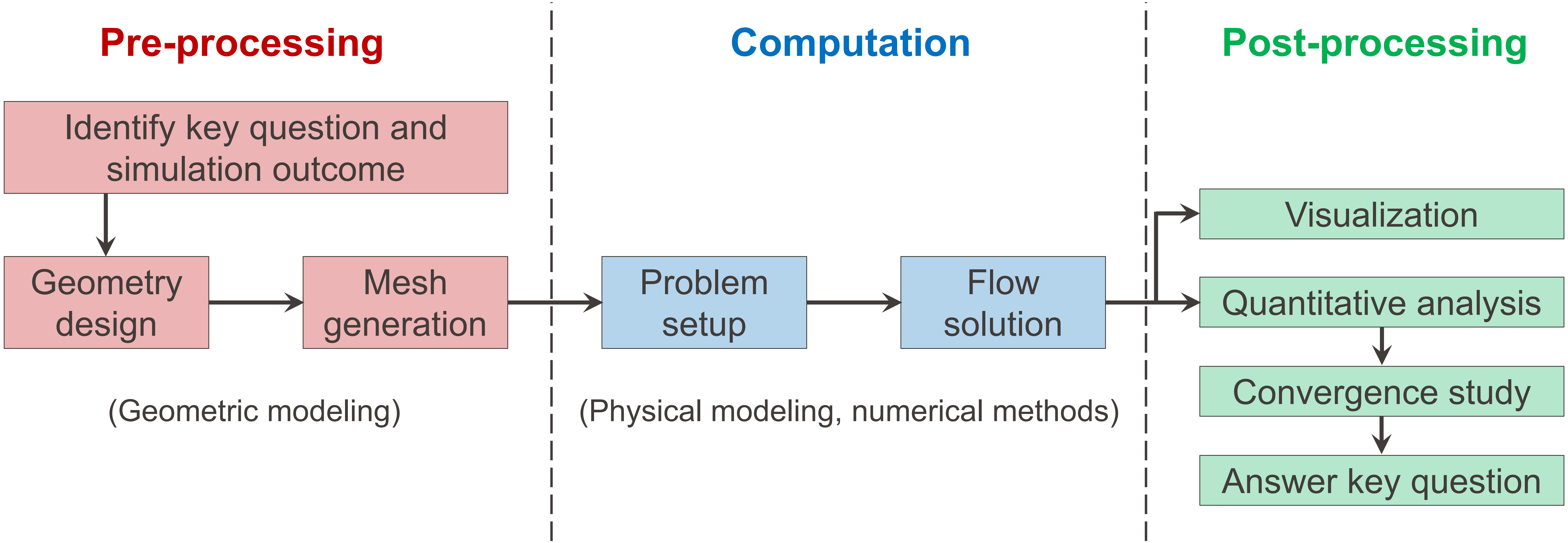
Discretization/computation methods

Some software
(Commercial – Free/open)

- Finite difference method (FDM) — (Mostly academic)
- Finite element method (FEM) — Comsol, FreeFEM++, AVBP…
- **Finite volume method (FVM)** — Fluent, Star-CCM+, OpenFOAM…
- Spectral elements — Nek5000…
- Spectral methods — (Mostly academic)
- Lattice Boltzmann methods (LBM) — PowerFLOW…
- Particle-based methods (SPH, DEM…) — SPH-flow, Fluidix…

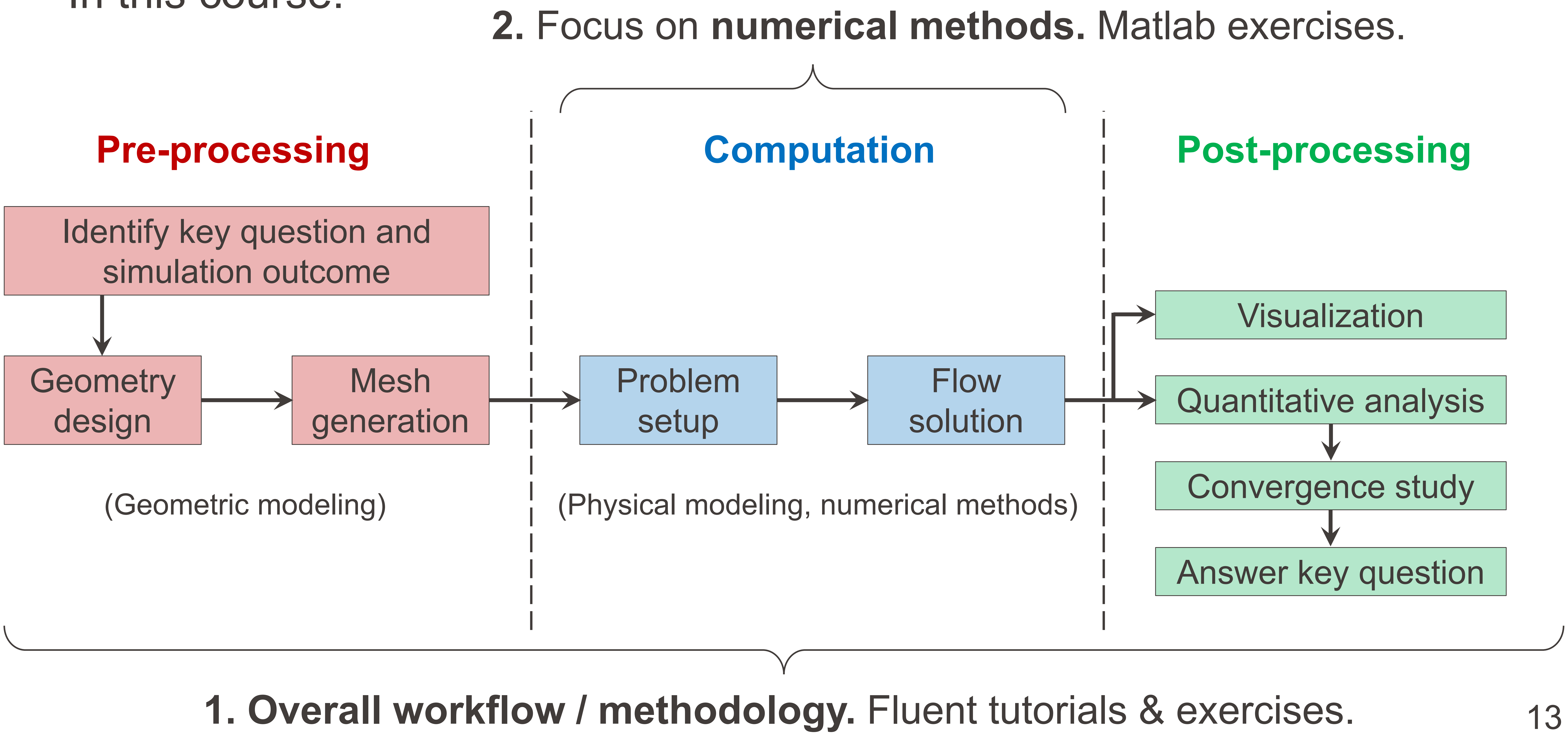
Numerical simulation workflow

- Workflow common to all software/methods:



Numerical simulation workflow

- In this course:



Objectives of this course

- Understand the **methodology** (different steps of the simulation workflow) and practice simulation techniques.

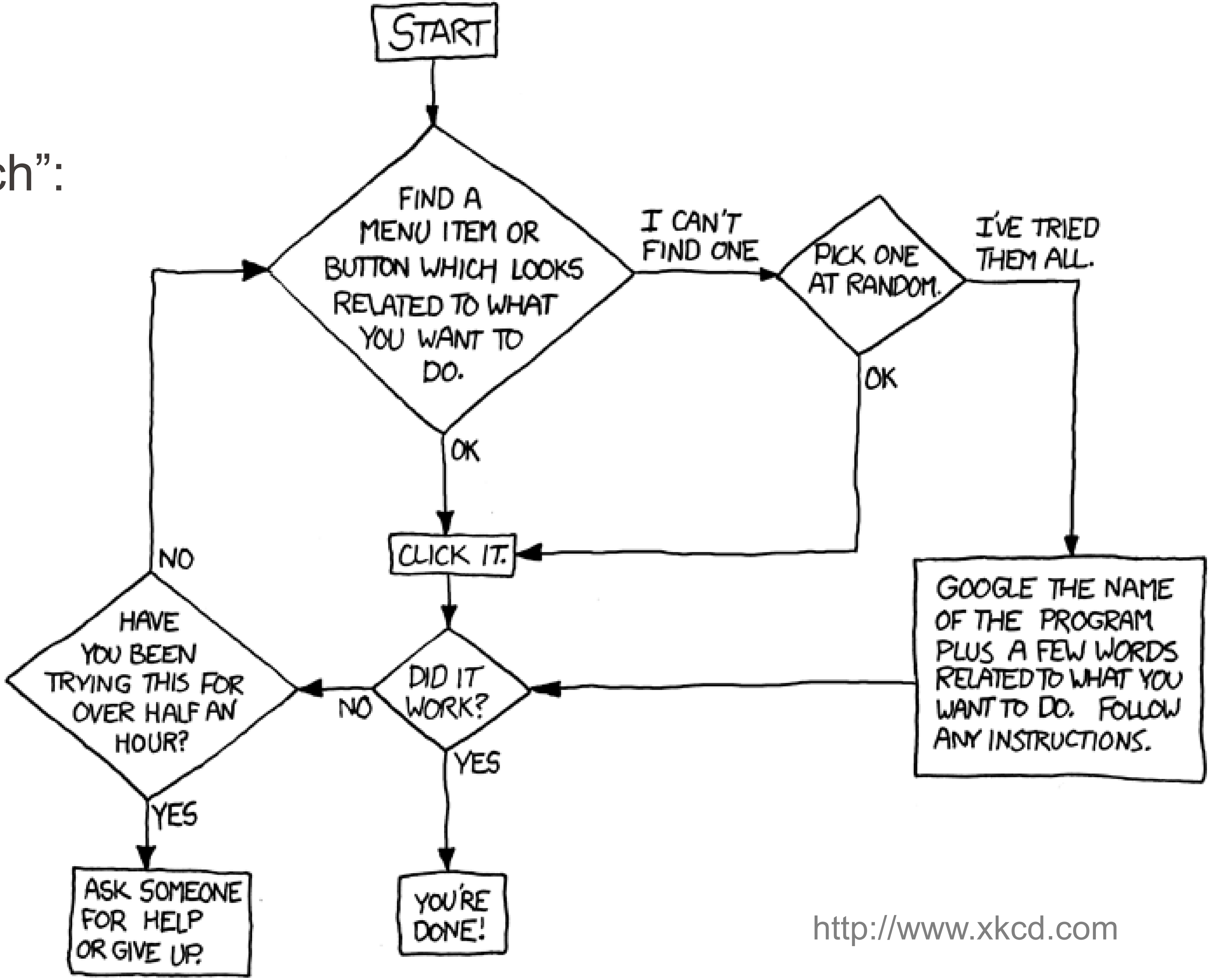
→ Become able to run a full simulation in an autonomous, structured and rigorous way.

- Understand some of the **numerical methods** involved.

→ Become able to make informed modeling / setup choices, and to justify them.

Objectives of this course

- Go beyond the "black-box approach":



http://www.xkcd.com

Fluid flow: notoriously difficult equations

- The Navier-Stokes equations are difficult to solve. Quizz: why?

$$\begin{cases} \nabla \cdot \mathbf{u} = 0 \\ \dfrac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p - \dfrac{1}{Re}\nabla^2 \mathbf{u} \end{cases}$$

- Compare for instance with the (much simpler) steady heat equation:

$$\nabla^2 T = 0$$

Fluid flow: notoriously difficult equations

- Some elements that make the Navier-Stokes equations difficult to solve:

$$\begin{cases} \nabla \cdot \mathbf{u} = 0 \\ \dfrac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p - \dfrac{1}{Re}\nabla^2 \mathbf{u} \end{cases}$$

Differential operators

Coupled equations

Unsteadiness

Nonlinearity

Pressure

Turbulence

- Additional complexity:
  - Multiphysics: chemical reaction, heat transfer, magnetohydrodynamics, fluid-structure interaction…
  - Multiphase: bubbles, drops, particles…

General information

- 5 credits

- Time repartition:
  - Lectures (2 h/week)
  - Exercise sessions: practical computer tutorials/exercises (2 h/week)
  - Personal work: exercises, group projects (about 2-3 h/week)

- Approach:
  - Combination of theory, demonstrations, tutorials and exercises
  - Combination of self-written Matlab codes (numerical methods, simple flows) and commercial CFD software Fluent (overall methodology, complex flows; NOT a Fluent training course).

Practical organization

- Lectures: 9:15-11:00 in INJ218. Previous years videos on SWITCHtube.
- Exercises: 11:15-13:00 in INJ218 (no computer) + BC07/08 (computers).

- Assistants: A. Amrani, J. Boscariol, C. Michaud-Lavoie, G. Zennaro (AEs) and A. Lagwankar, M. Mohammadrashidi (TAs from LFMI)
- Support from T. Favre, scientist at LFMI, formerly at Caterham F1, Lotus F1, Renault Sport, Sauber Motorsport, and Mercedes-AMG Petronas.

- All announcements / documents on Moodle.

- Feedback about the course welcome at any time!

Practical organization

- Software: Matlab and Fluent
  - Available in computer rooms.
  - Available on virtual machines “STI-WINDOWS11” via VDI (Virtual Desktop Infrastructure, vdi.epfl.ch). More info on Moodle.
  - Can be installed on your personal computers (Matlab from Distrilog, Fluent student version from ANSYS website). Please check conditions of use.

- Assessment:
  - 2 project reports: 1 Matlab + 1 Fluent (details in next slides),
  - 1 written exam.

Assessment: mini-projects

- 2 mini-projects:
  - 1 with Matlab (focused on numerical methods).
  - 1 with Fluent (focused on overall workflow / methodology).

- Objective: apply the course contents to a “practical” problem.

- Organization:
  - In parallel of the course, during the **whole semester** (Matlab first, then Fluent).
  - **Groups of 4 students**. Please fill in the **list on Moodle**.

Mini-projects

- **Workload**: total of about 30 h/person (outside contact hours).

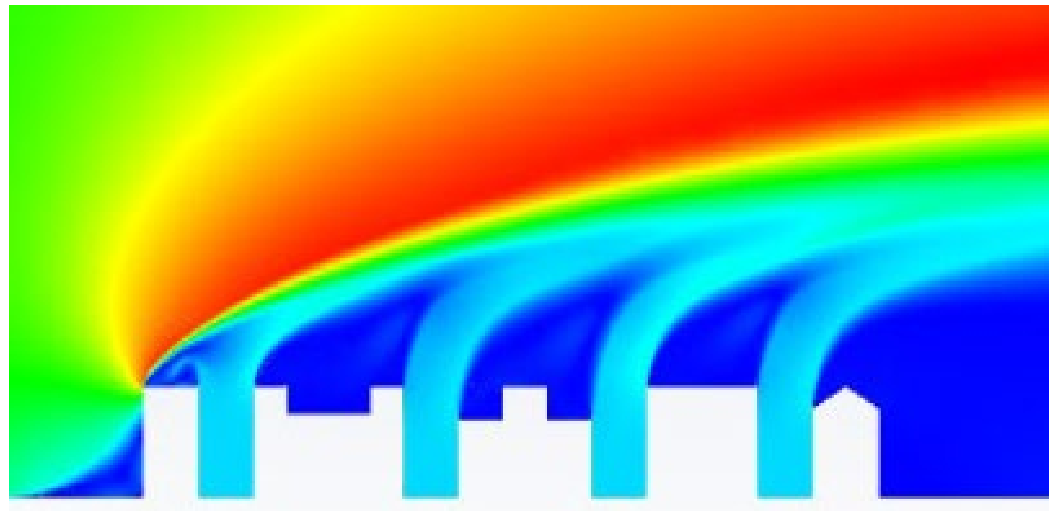
- **Evaluation** based on the **final reports** only.

- **Work equitably distributed** among group members, **same grade to all members.**

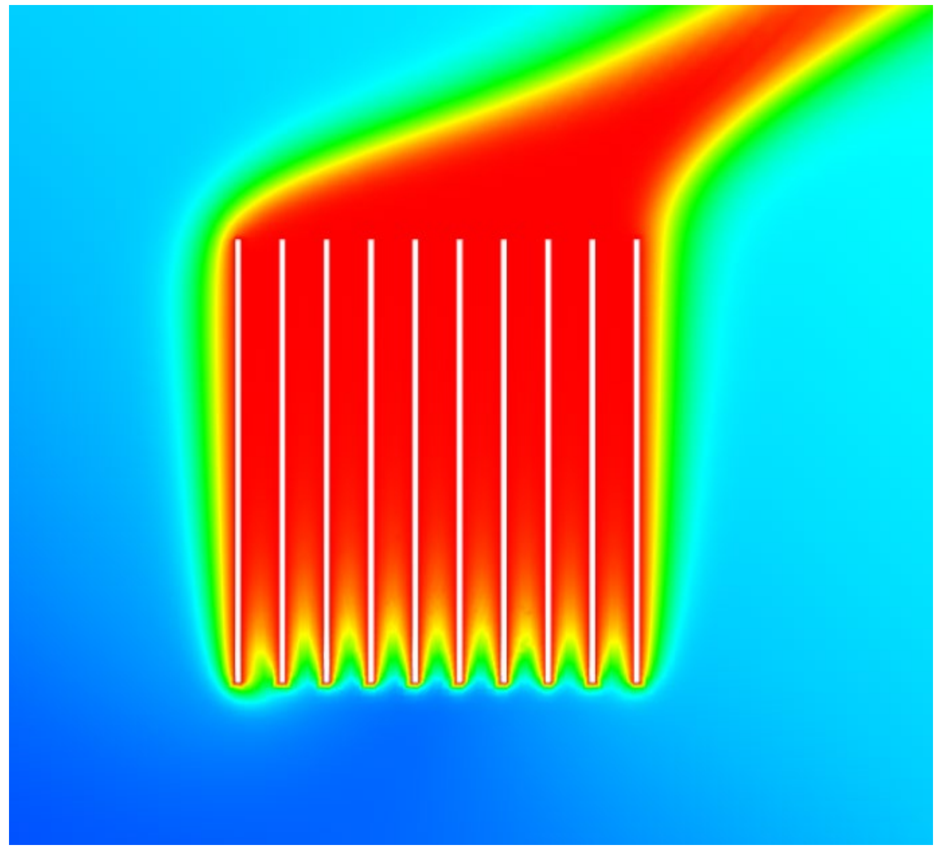
- **Topic**:
  - Matlab: one single subject (same for all groups).
  - Fluent: chosen by you (may be related to other course/project, personal interest/hobby etc.), or among a list of propositions if you don't have ideas.

Fluent mini-projects: examples

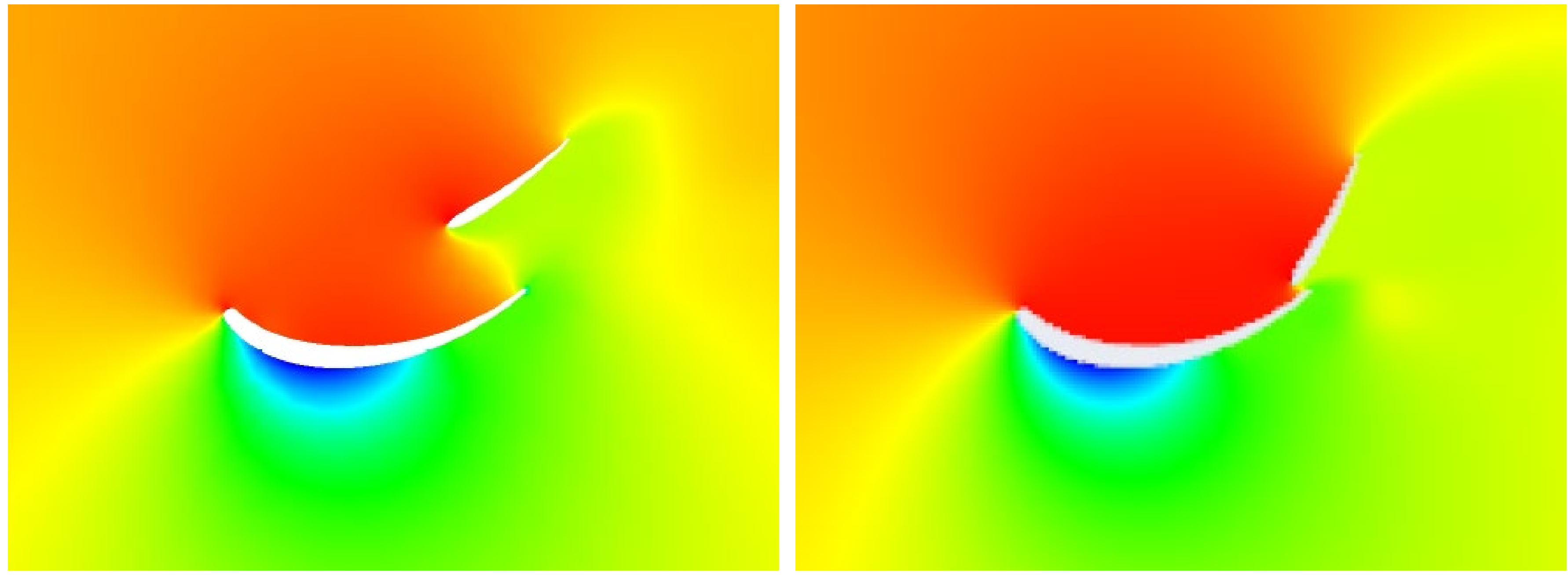
Urban pollution



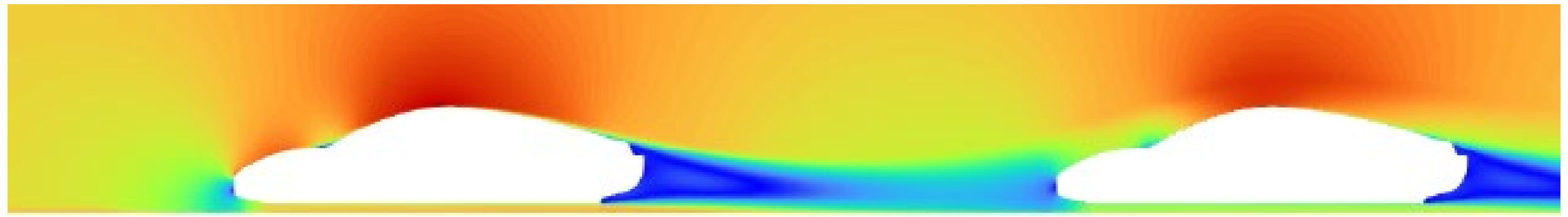
Heat exchanger



F1 rear wing

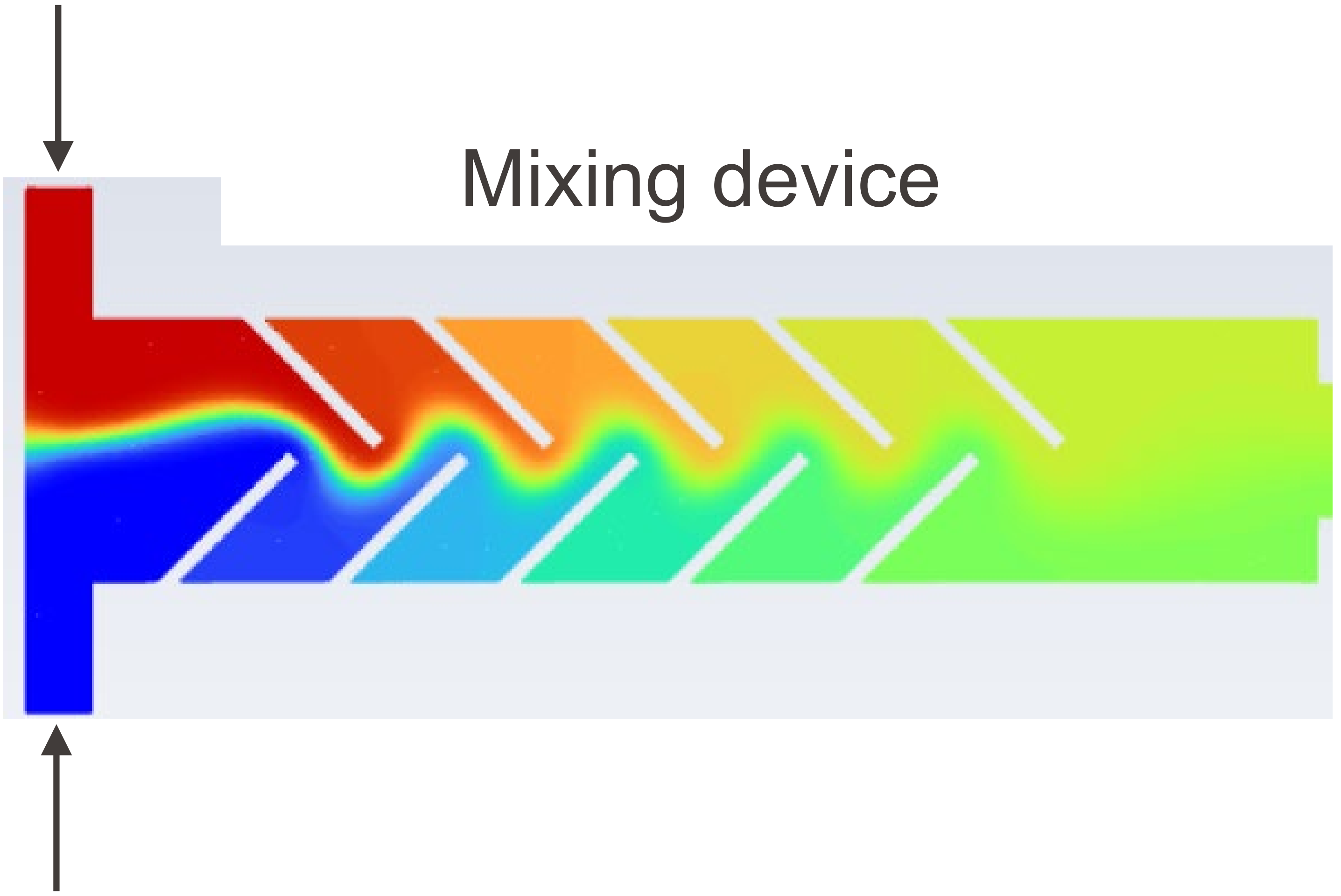


Car slipstreaming

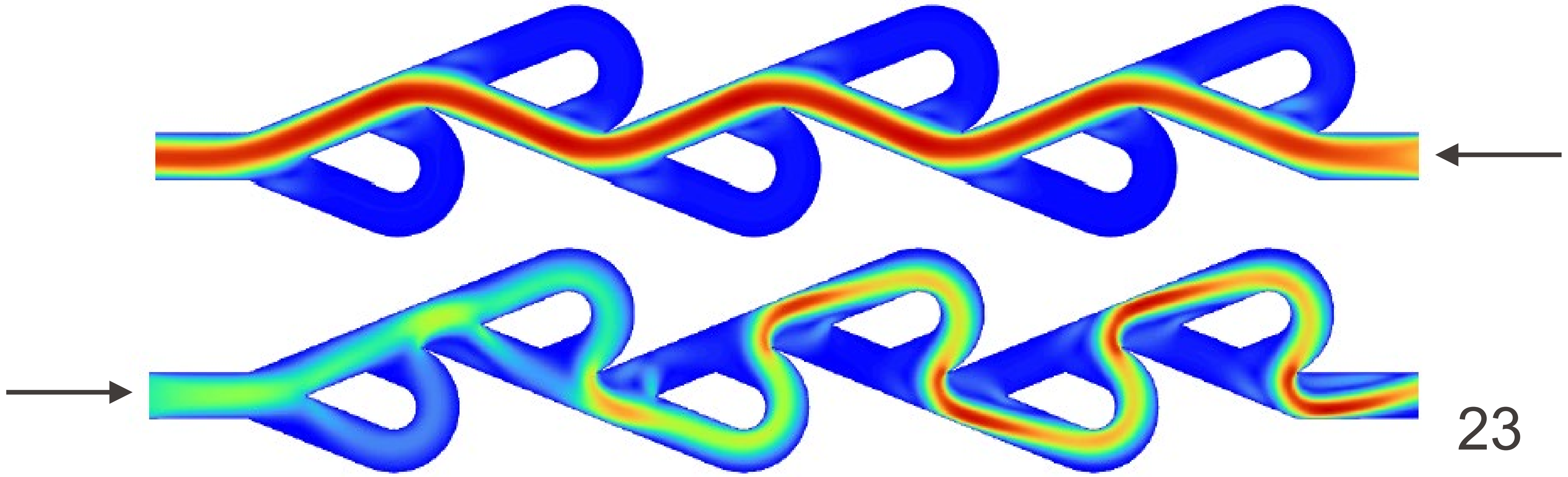


Numerical Flow Simulation

Mixing device



Tesla valve (fluidic diode)



Fluent mini-projects: some rules to follow

1. Computational mesh ≤ 500'000 elements → 2D or axisymmetric flow. No 3D.

2. Stick to simple physics, avoid complex multi-physics:
   - No solid thermal conduction. (But fluid thermal convection-diffusion OK.)
   - Fluid-structure interaction: no 2-way coupling. (But 1-way coupling OK for fluid flow with prescribed solid displacement).
   - Two-phase flow OK to some extent.
   - No combustion, no chemical reactions, no phase change (evaporation, solidification etc.), no aero-acoustics…

3. No optimization. (But reasonable parametric studies OK.)

Learning prerequisites

- Required courses:
  - Fluid mechanics (for ex. ME-280)
  - Numerical analysis (for ex. MATH-251)

- Recommended courses:
  - Fluid flow (ME-271)
  - Discretization methods in fluids (ME-371)

- Other useful concepts/skills:
  - Matlab, Python or equivalent
  - CAD

References

- Books:
  - *Numerical heat transfer and fluid flow*, S. Patankar (1980)
  - *An introduction to Computational Fluid Dynamics – The Finite Volume Method*, H.K. Versteeg, W. Malalasakera (2007)
  - *Computational methods for fluid dynamics*, J.H. Ferziger, M. Perić (1999)

- Software documentation:
  - Matlab: help
  - ANSYS Fluent: user guide, theory guide. Extensive and very useful.