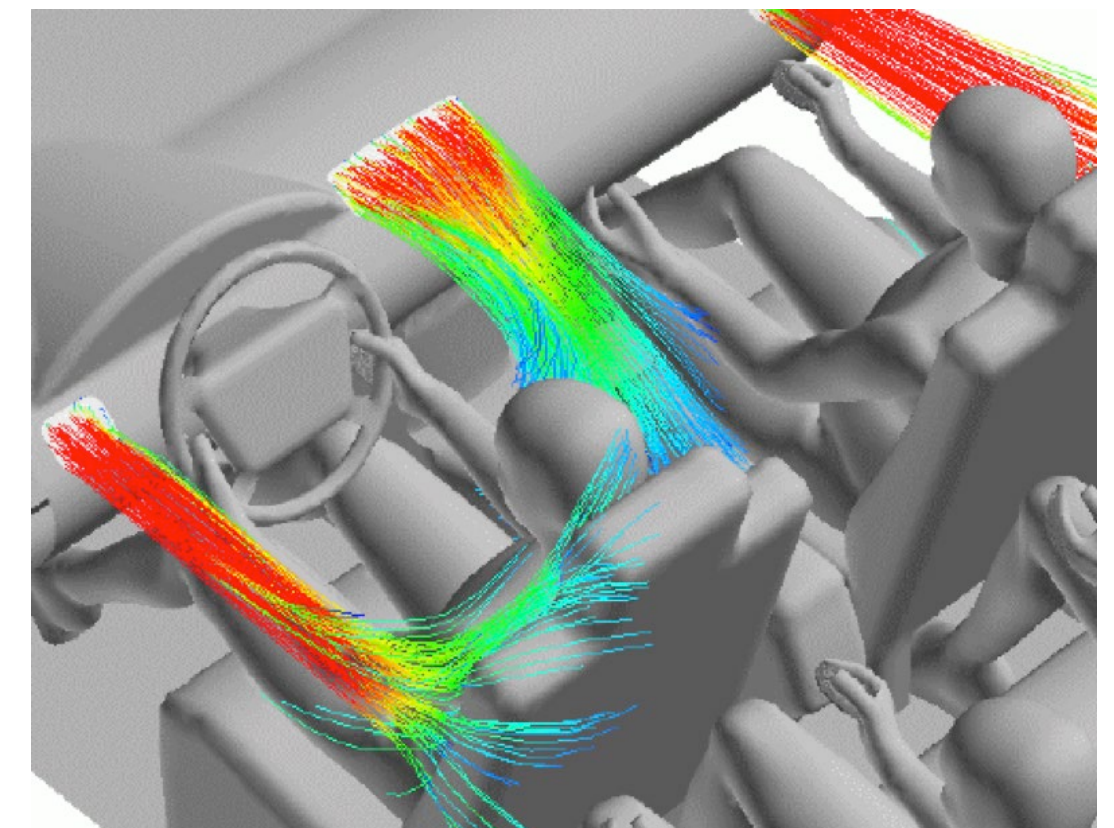
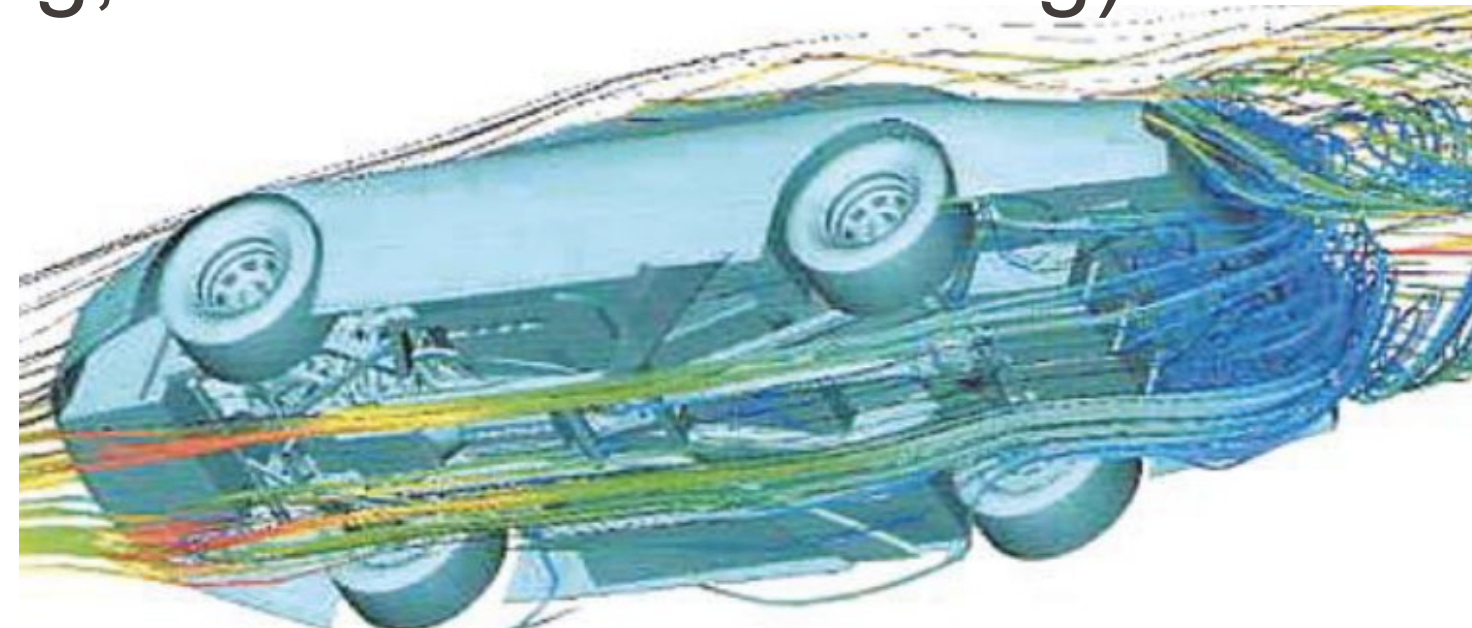
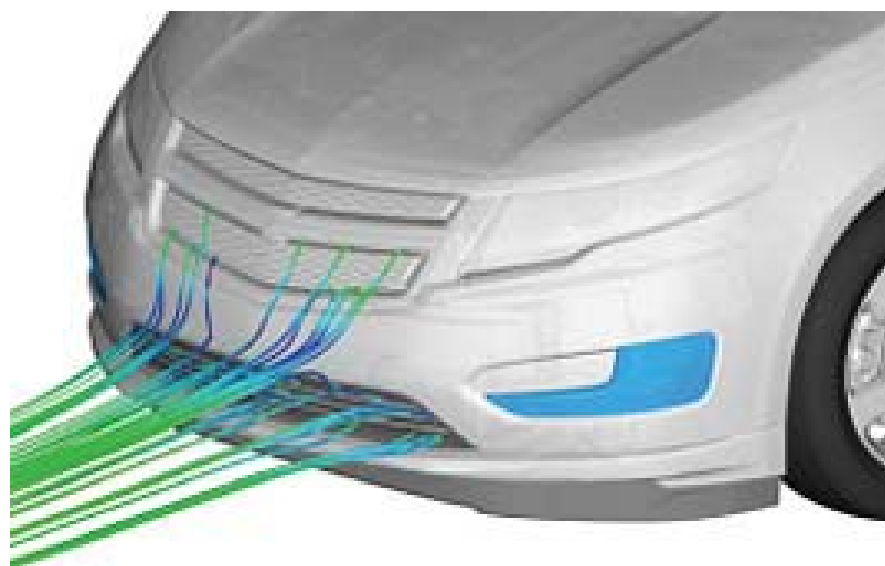
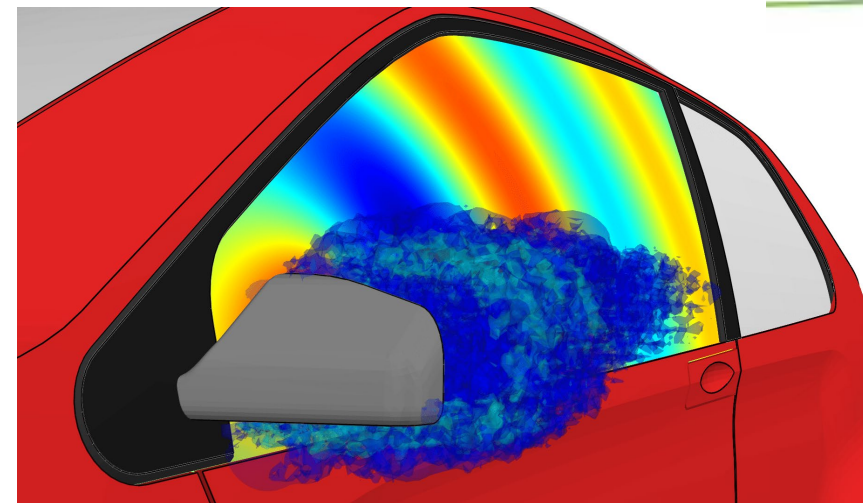
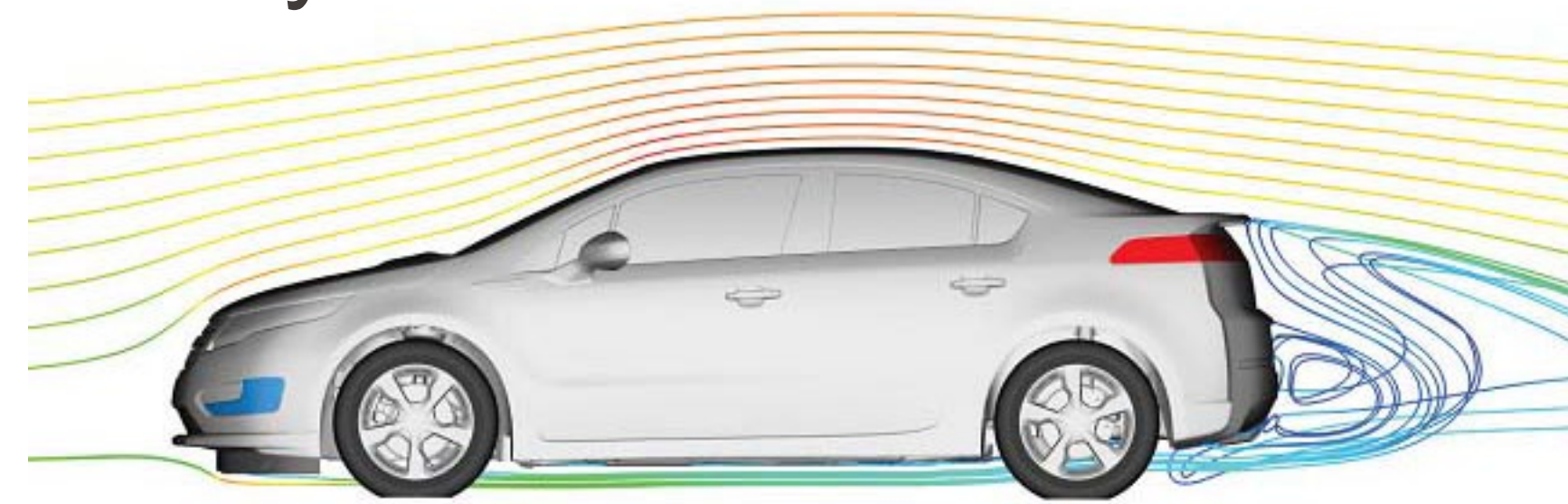


Introduction

Numerical Flow Simulation

Self-introduction

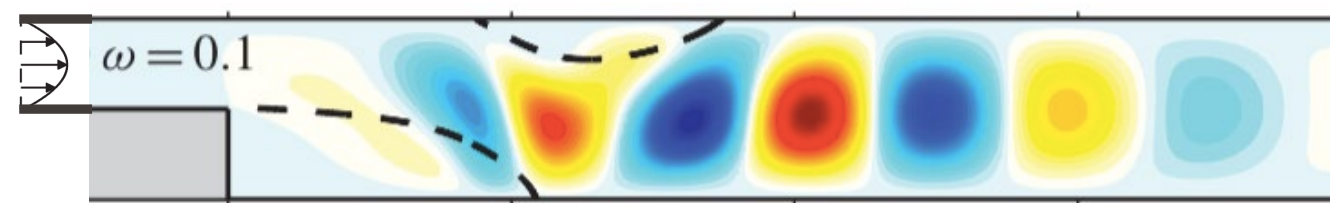
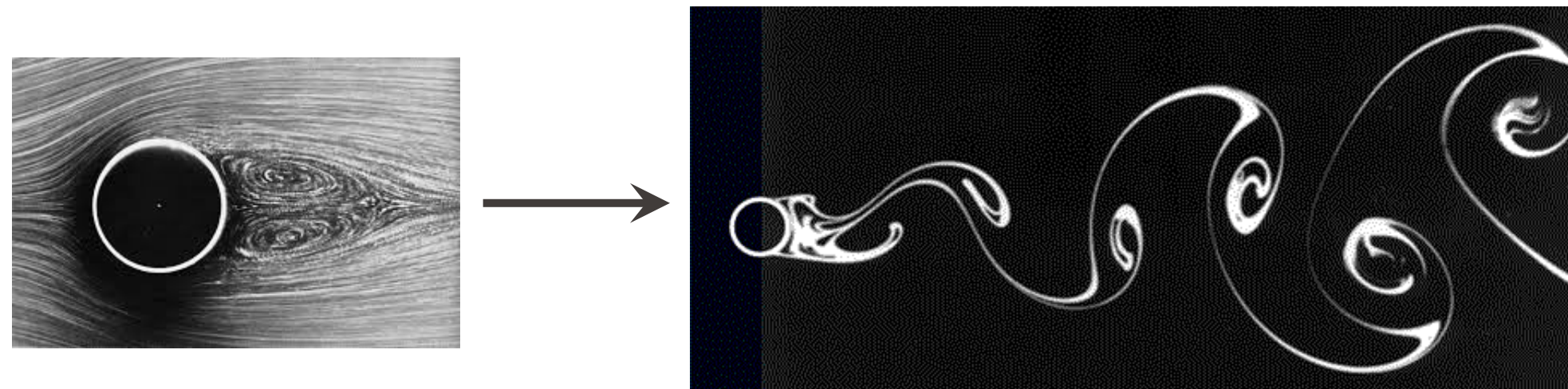
- Previously: CFD engineer in the automotive industry
- Method development for
 - Aerodynamics: drag (fuel consumption), lift (handling & stability),
 - Aeroacoustics (“wind noise”),
 - Thermal management (engine cooling, underhood & underbody cooling, cabin air conditioning)



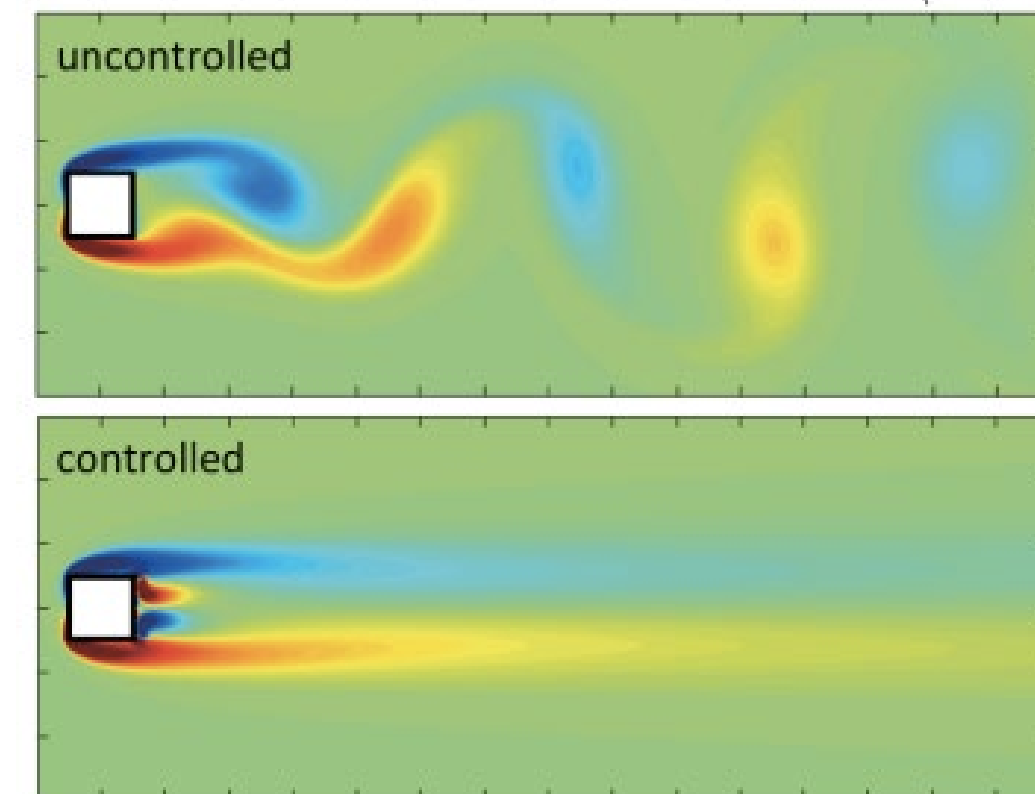
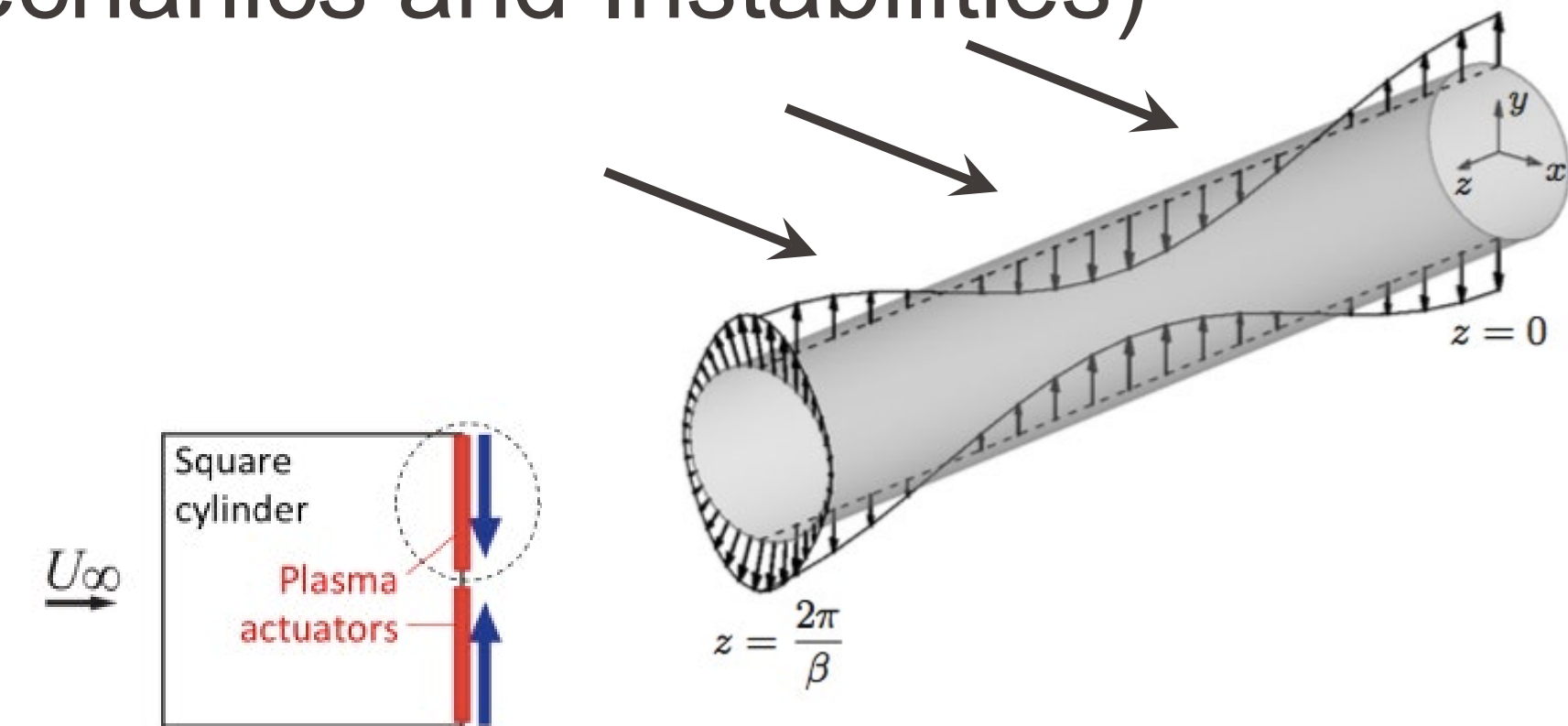
Self-introduction

- 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?

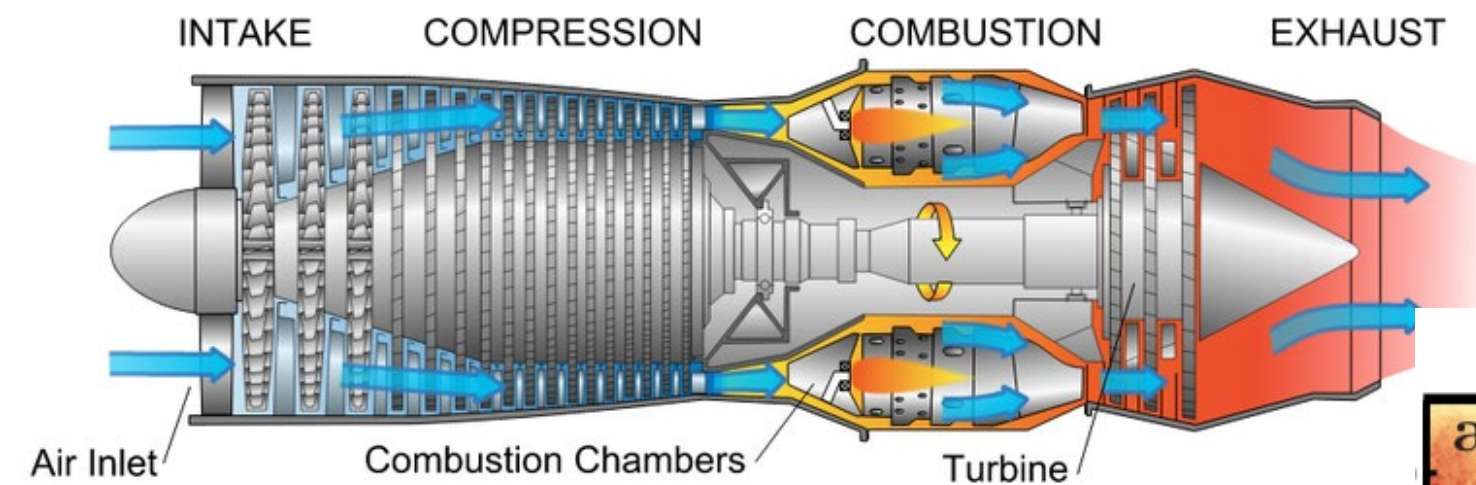


- Systematic methods (optimal control)
to improve stability or aerodynamics

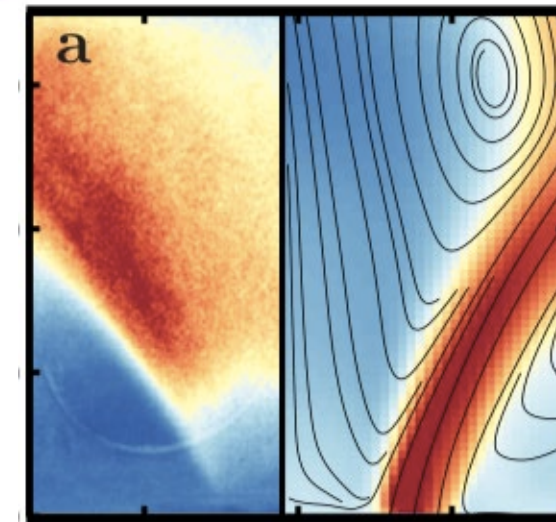
Self-introduction

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

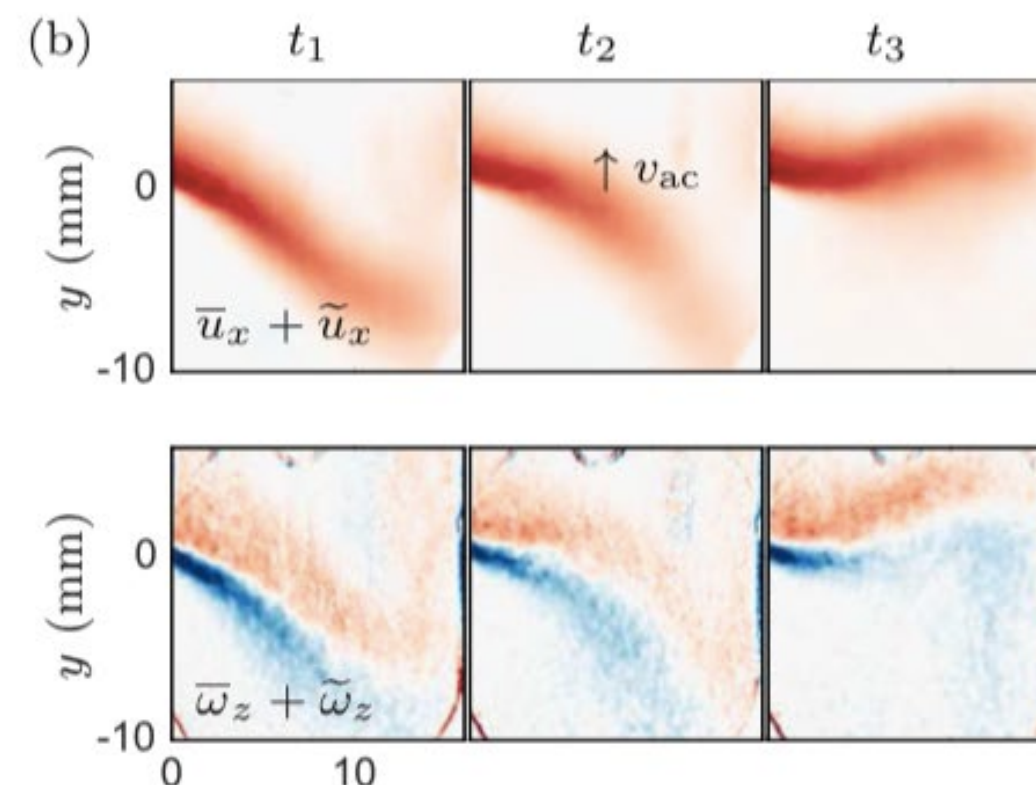
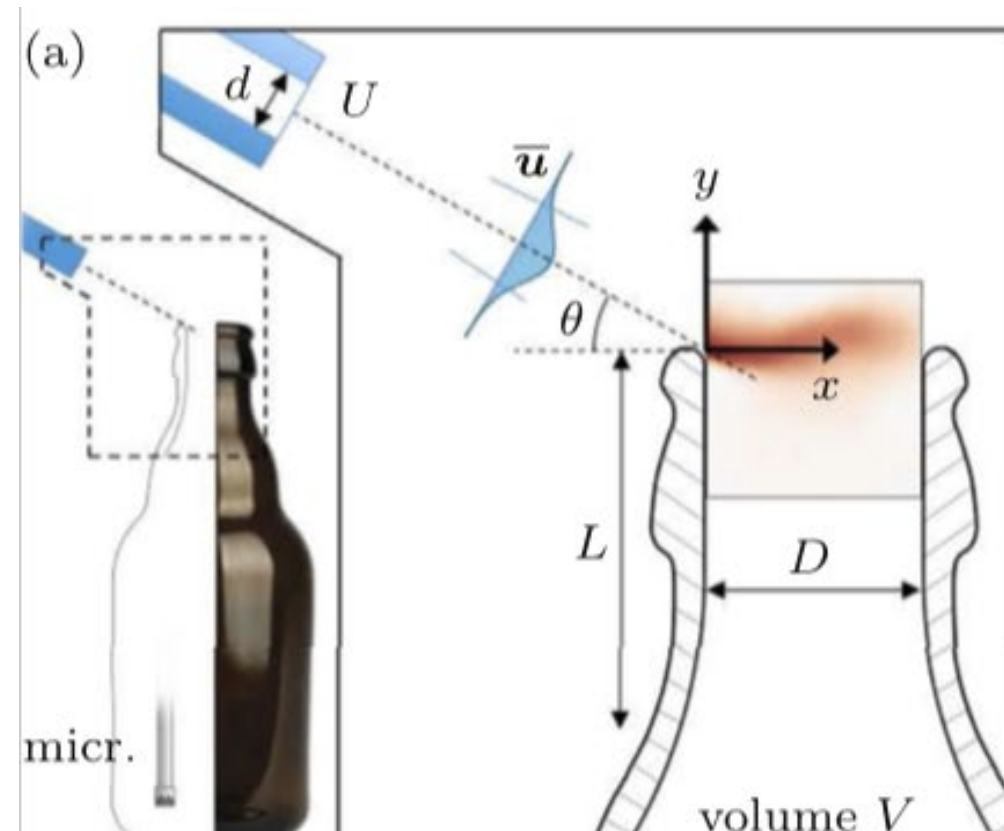
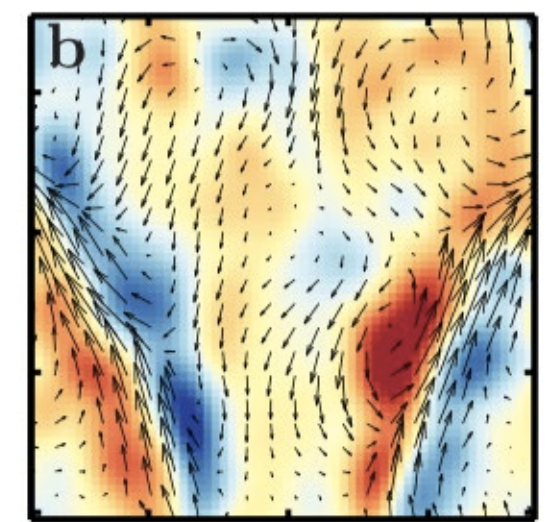
- Research activities:
 - Flow stability and control
 - Thermo-/aero-acoustics**
 - Fluid-structure interaction
 - Thin-film coating



Averaged flow



Instantaneous flow



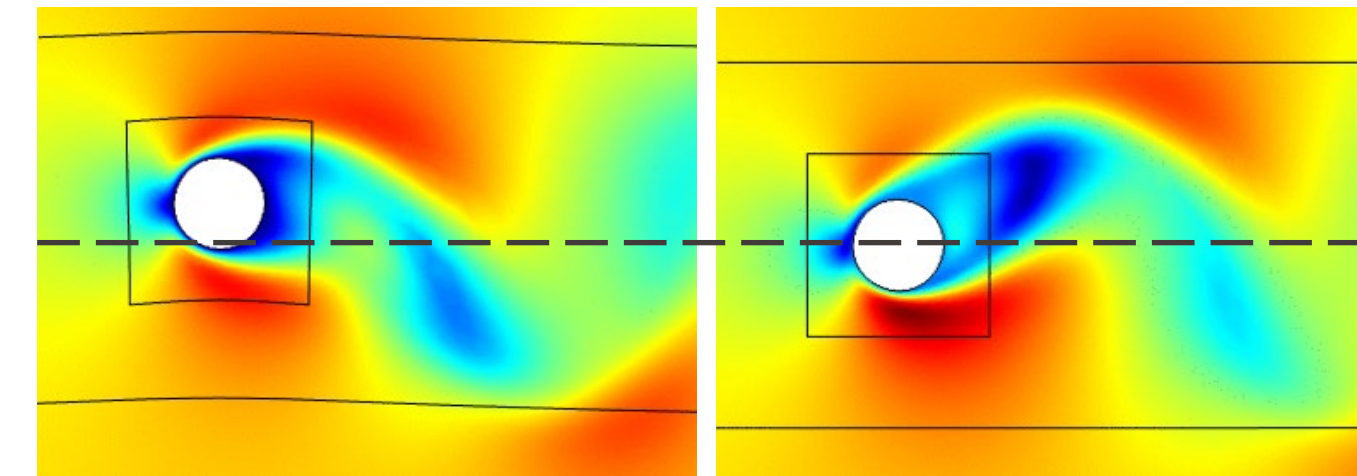
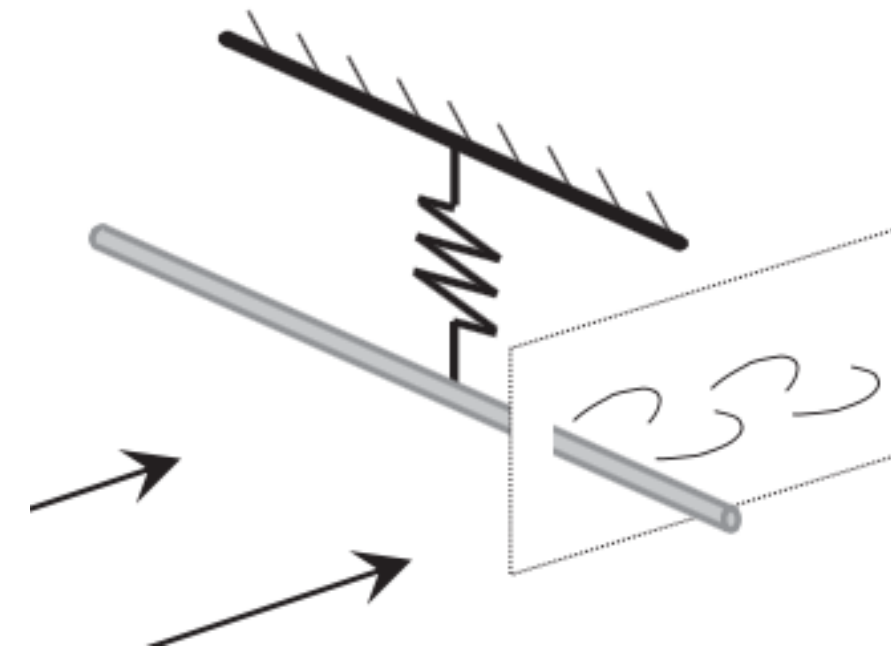
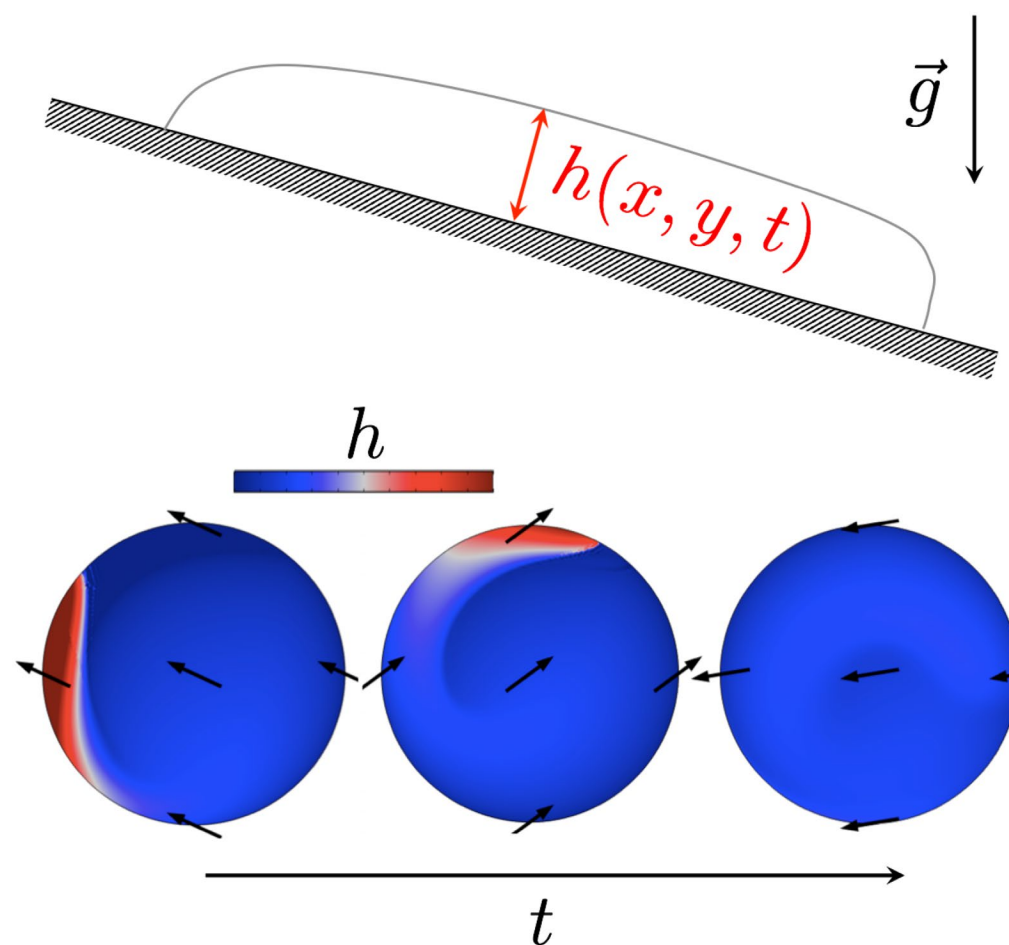
- Coupling between oscillations of pressure and flame, or pressure and jet/mixing layer
- Low-order modelling and identification of instabilities in turbulent systems

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**

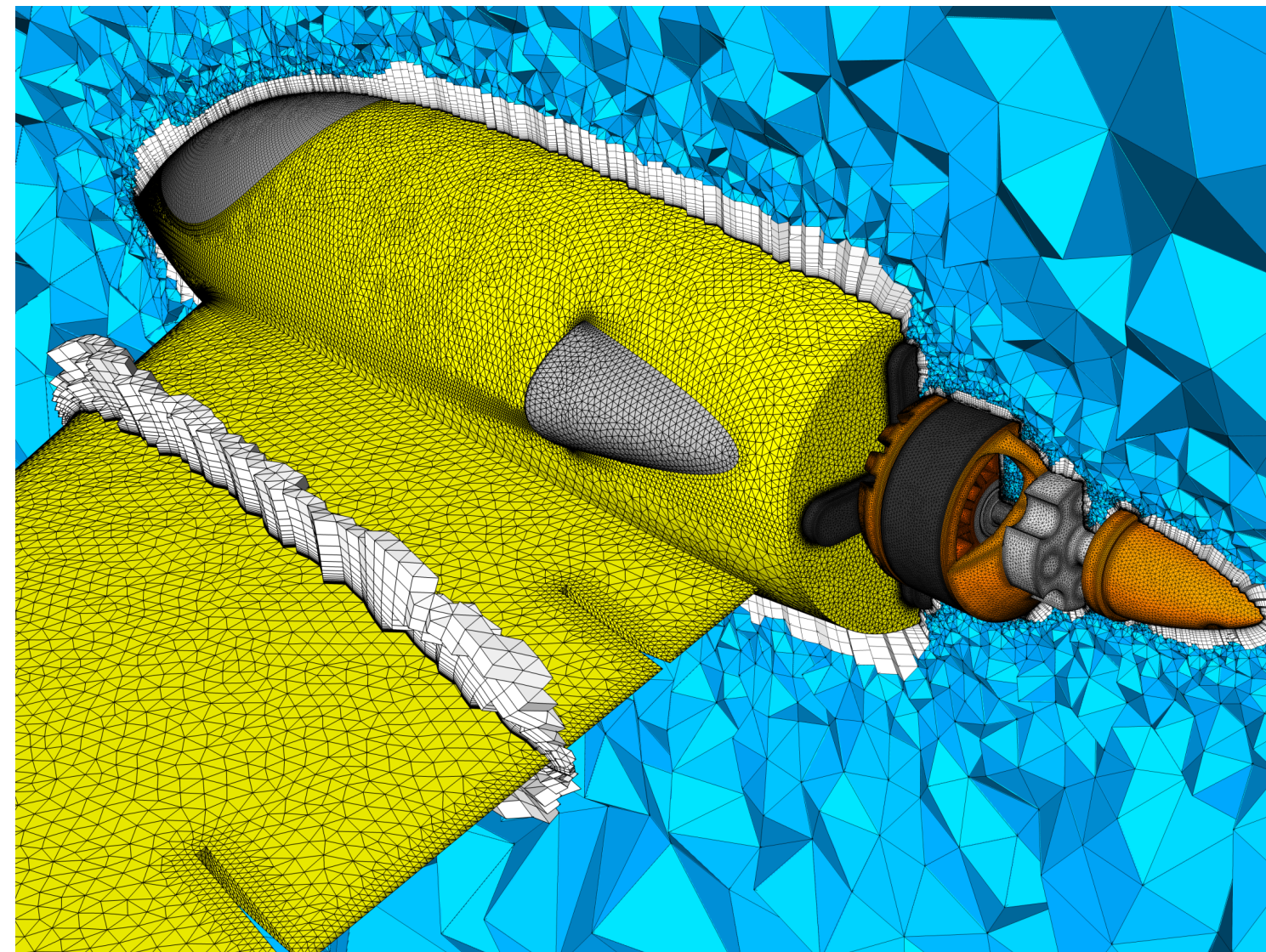


- Energy harvesting, stability properties, low-order modelling

- Optimal motion for uniform coating

Numerical flow simulation

- A.k.a. “Computational Fluid Dynamics” (CFD)
- General idea: discretize fluid domain $\rightarrow$ transform governing equations (PDEs for continuous variables) into approximate equations (algebraic equations for discrete variables) $\rightarrow$ solve numerically (computer)



Numerical flow simulation

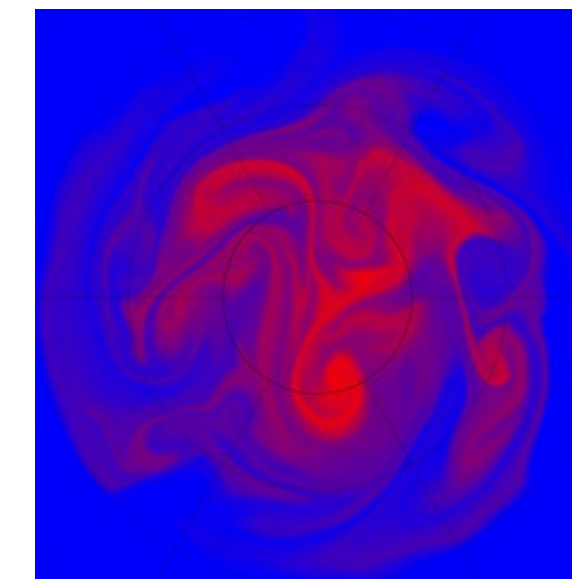
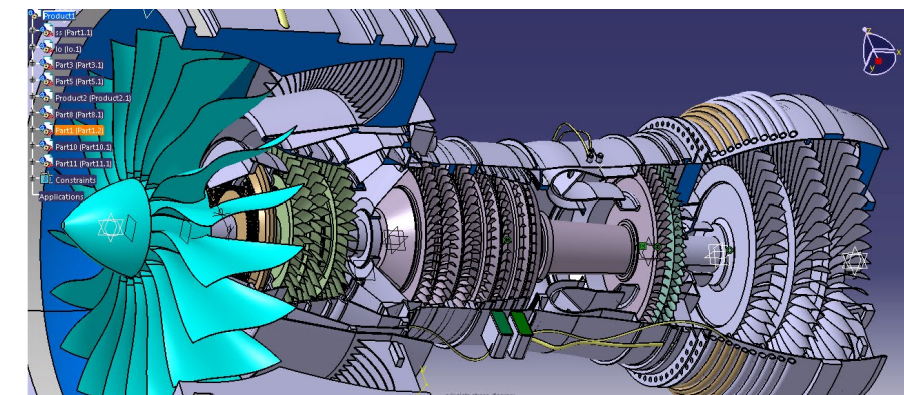
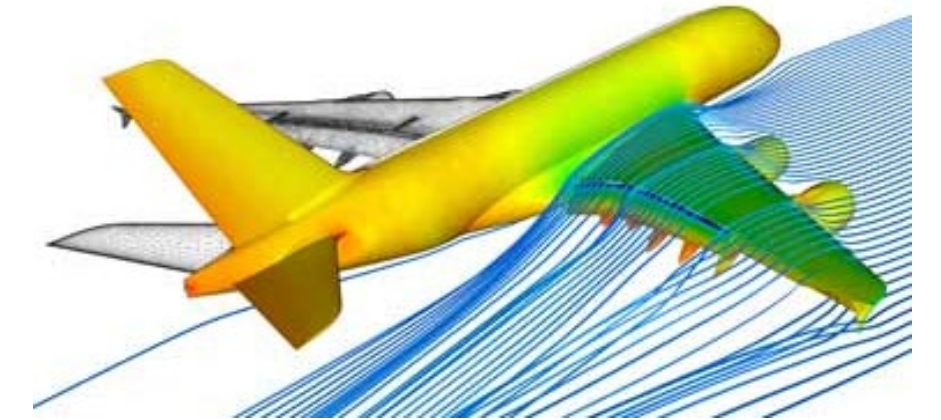
- Advantages:
 - Can handle complex equations/geometries not solvable analytically
 - Useful when experiments are expensive, difficult, dangerous, time consuming
 - Complete information (time and space-resolved)
 - Parametric studies (+ optimization)

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

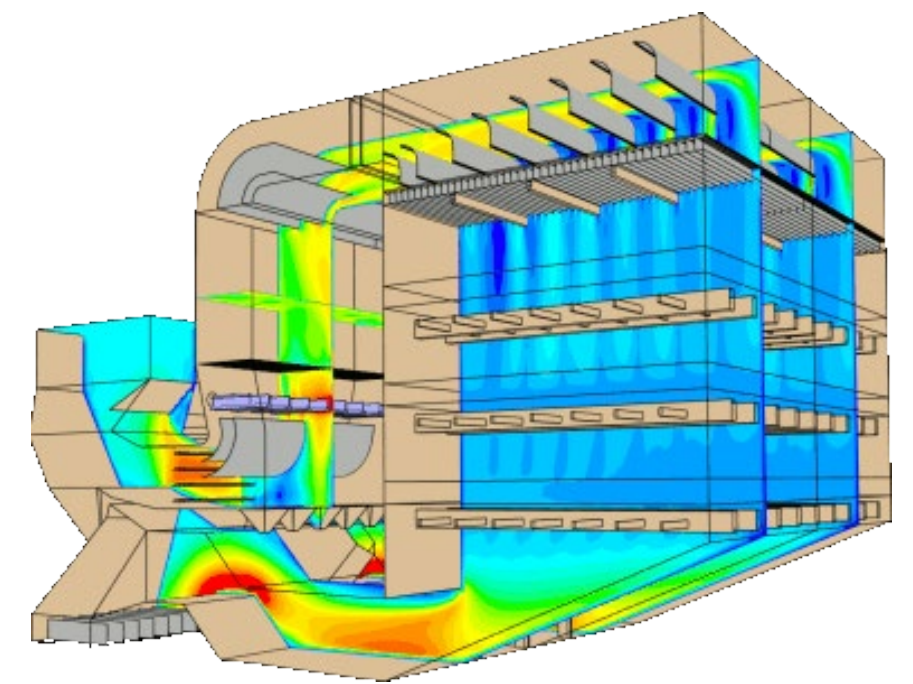
- Broad range of applications in research and engineering:
 - Aerodynamics, aeroacoustics (aircraft, ground vehicles)
 - Ship hydrodynamics
 - Engines
 - Energy: gas turbines, wind turbines, hydro/nuclear power plants
 - Food & chemical processes: foam, emulsions, mixing, particle transport, polymer molding



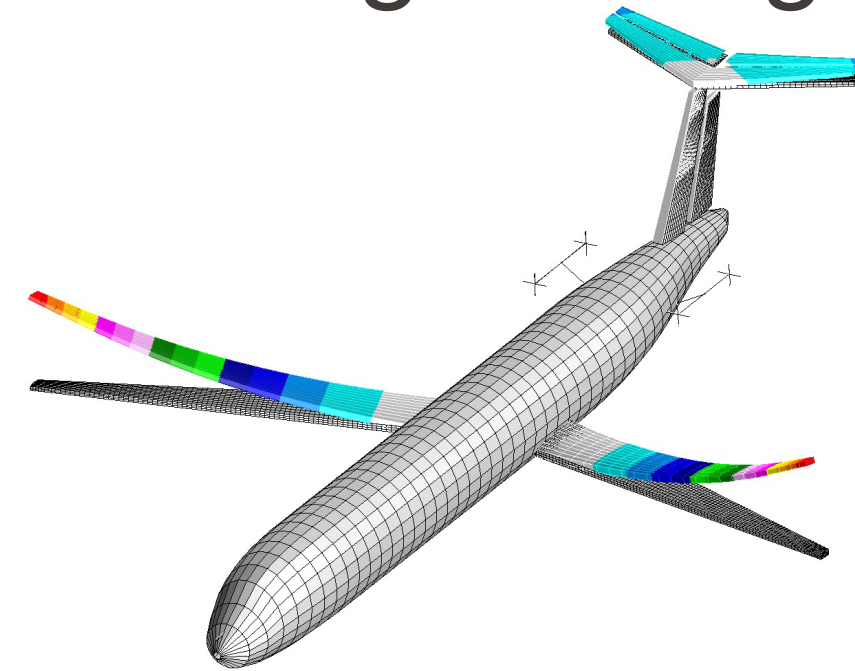
Numerical flow simulation

- Broad range of applications in research and engineering:

- Building thermal management



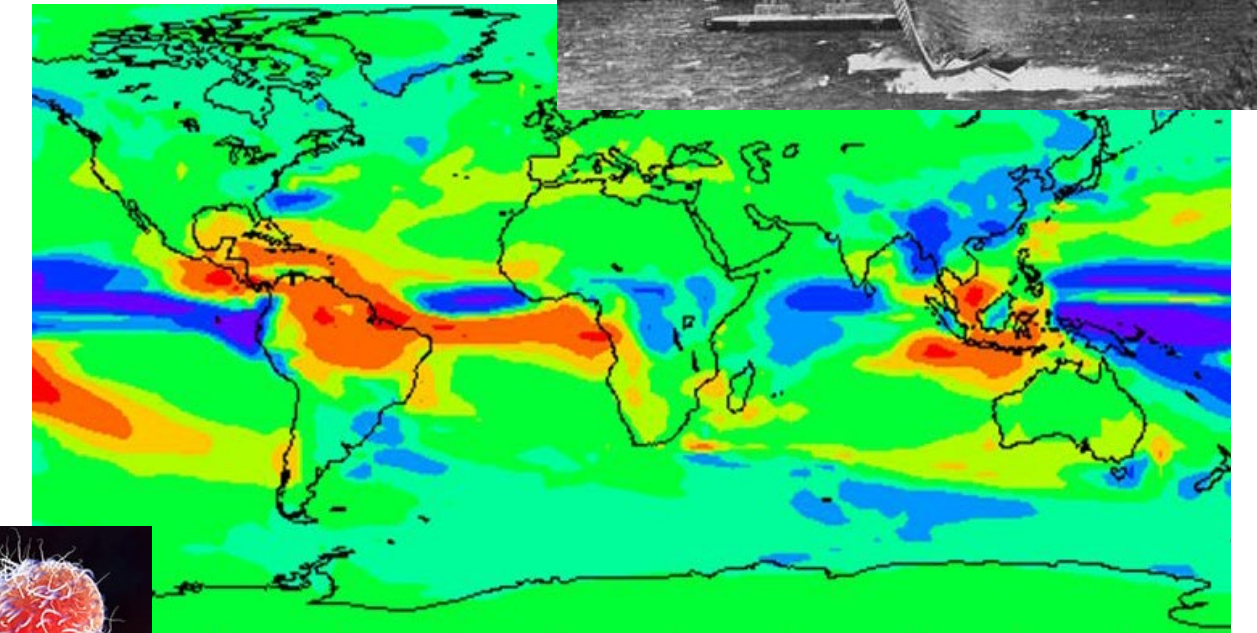
- Fluid-structure interaction: aeroelasticity, civil engineering, energy harvesting



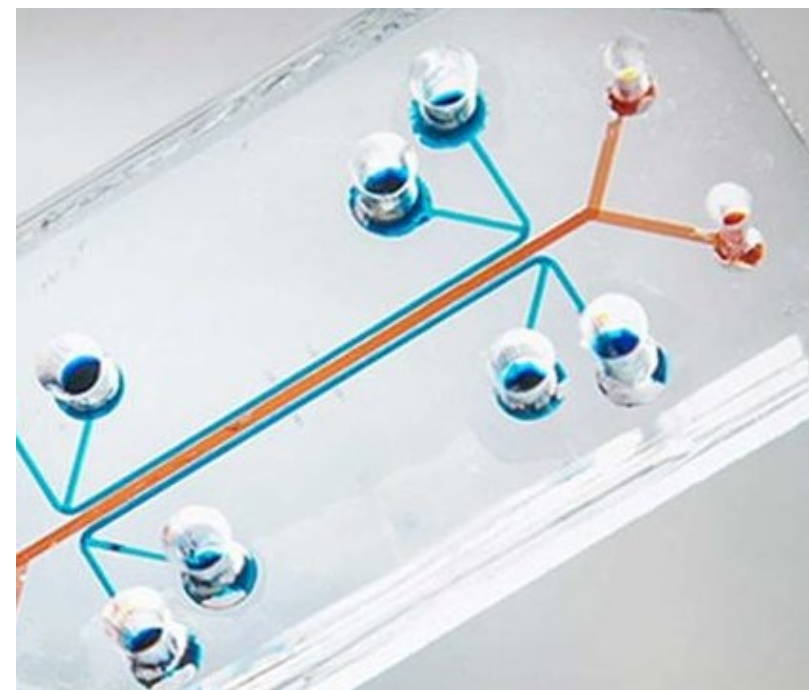
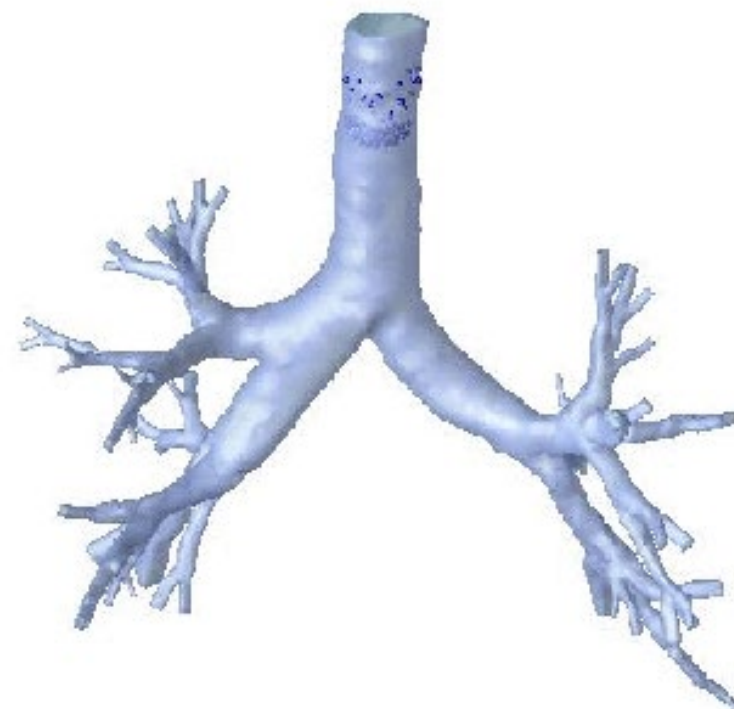
- Meteorology: weather prediction, global climate



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



- ...



Discretization/computation methods

Some software
(Commercial – Free/open)

- Finite difference method (FDM)
- Finite element method (FEM)
- **Finite volume method (FVM)**
- Spectral elements

(Mostly academic)

Comsol, FreeFEM++, AVBP...

Fluent, Star-CCM+, OpenFOAM...

Nek5000...

- Spectral methods
- Lattice Boltzmann methods (LBM)

(Mostly academic)

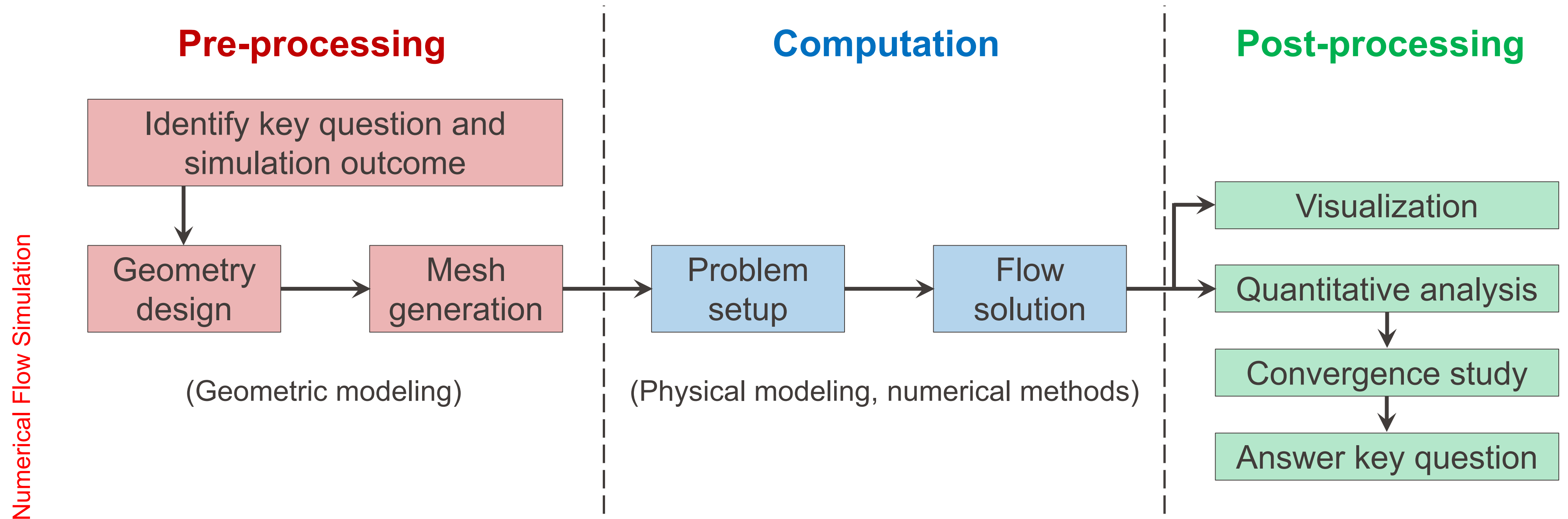
PowerFLOW...

- Particle-based methods (SPH, DEM...)

SPH-flow, Fluidix...

Numerical simulation workflow

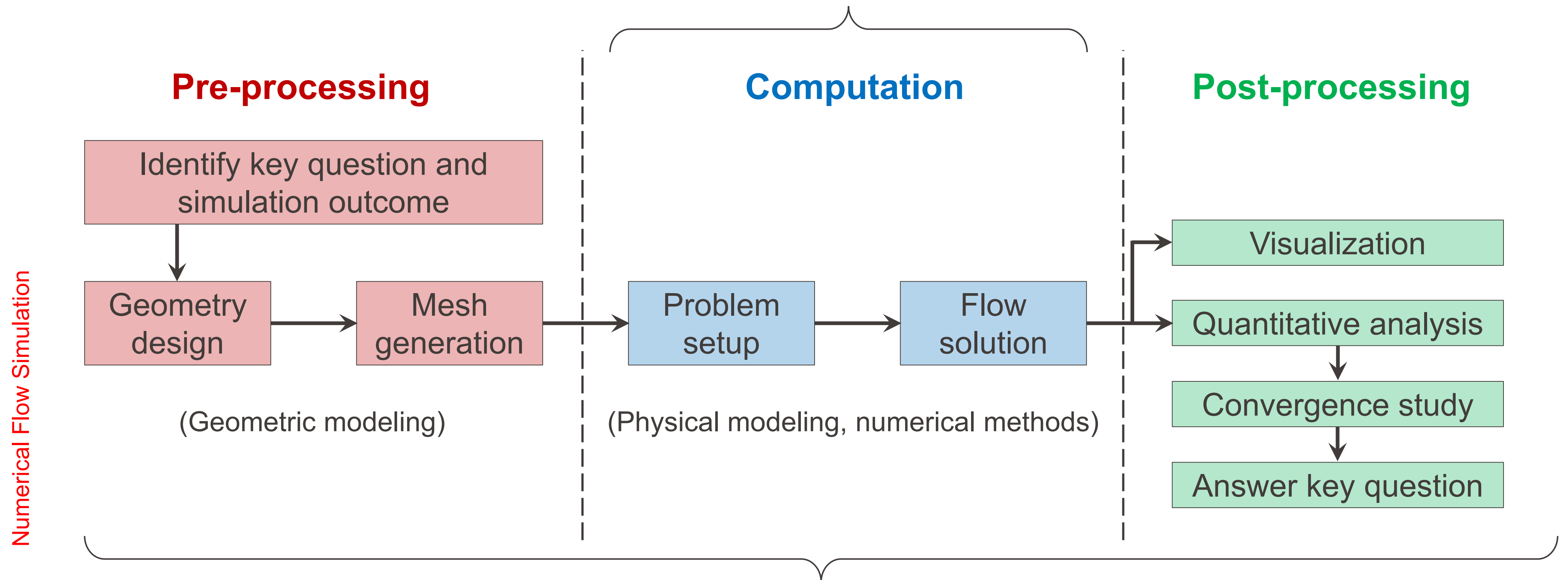
- Workflow common to all software/methods:



Numerical simulation workflow

- In this course:

2. Focus on **numerical methods**. Matlab exercises.



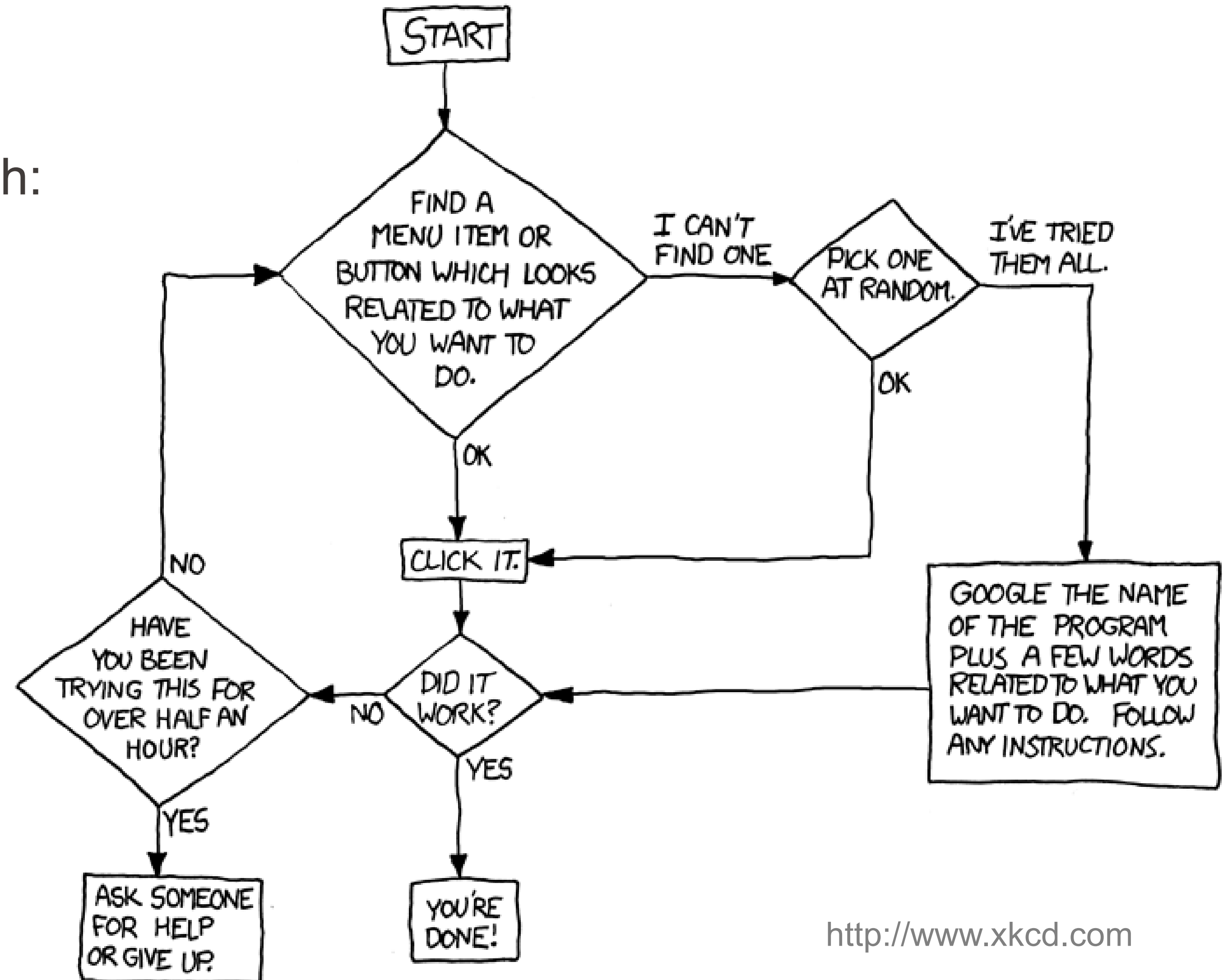
1. Overall workflow / methodology. Fluent tutorials & exercises.

Objective 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.

Objective of this course

- Go beyond a black-box approach:



Fluid flow: notoriously difficult equations

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

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

- 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:

The diagram shows the Navier-Stokes equations with annotations in red and blue. A red bracket on the left groups the equations as "Coupled equations". Blue lines point from "Differential operators" to the divergence and Laplacian terms. A blue line points from "Unsteadiness" to the time derivative term. A red line points from "Nonlinearity" to the convective term. A blue line points from "Pressure" to the pressure gradient term. A red line points from "Turbulence" to the Reynolds number term.

$$\left\{ \begin{array}{l} \nabla \cdot \mathbf{u} = 0 \\ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p - \frac{1}{Re} \nabla^2 \mathbf{u} \end{array} \right.$$

Annotations:

- Coupled equations** (red)
- Differential operators** (blue)
- Unsteadiness** (blue)
- Nonlinearity** (red)
- Pressure** (blue)
- Turbulence** (red)

- 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) + CM1103 (computers).
- Assistants: A. Silvestri, I. Beniffou, M. De Pascalis, V. Scurria (AEs) and K. Wittkowski (LFMI).
- All announcements / documents on Moodle.
- Feedback about the course welcome at any time!

Practical organization

- Software: Matlab and Fluent
 - Available on computers in room CM1103.
 - Available on virtual machines “STI-WINDOWS10” 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 (see details in next slides),
 - 1 written exam.

Assessment: mini-projects

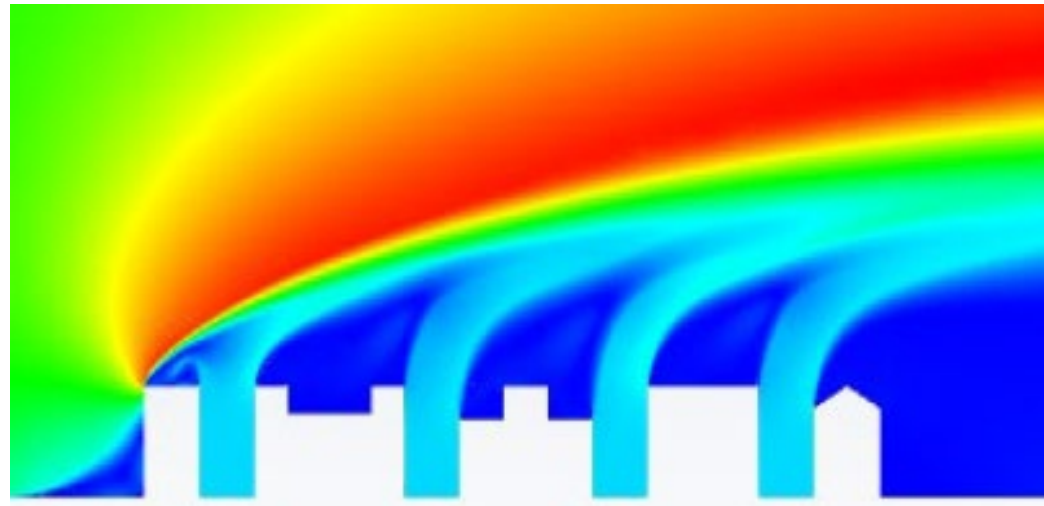
- 2 mini-projects:
 - 1 with Matlab (focused on numerical methods).
 - 1 with Fluent (focused on overall workflow / methodology).
- Objective: apply the course contents to a “practical” problem.
- Organization:
 - In parallel of the course, during the **whole semester** (Matlab first, then Fluent).
 - **Groups of 3-4 students.** Please fill in the **list on Moodle**.

Mini-projects

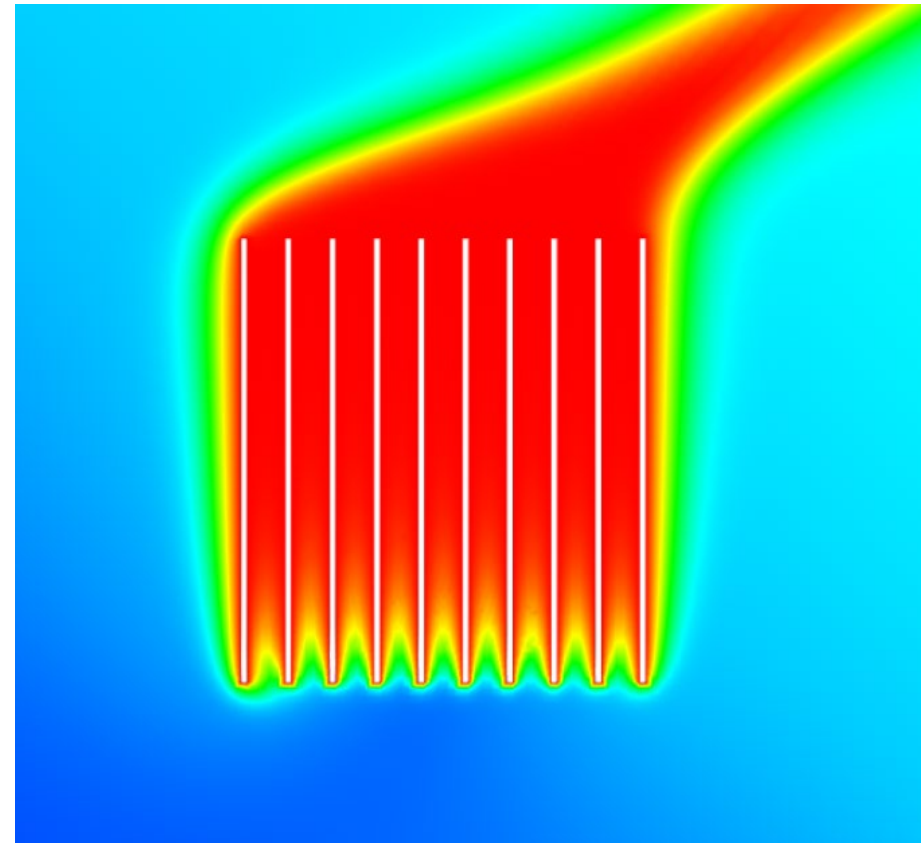
- **Workload:** total of about 30 h/person (outside contact hours).
- **Evaluation** based on the **final reports** only.
- **Work equitably distributed** among group members, **same grade to all members.**
- **Topic:**
 - Matlab: one single subject (same for all groups).
 - Fluent: chosen by you, or among a proposed list if you don't have ideas.

Fluent mini-projects: examples

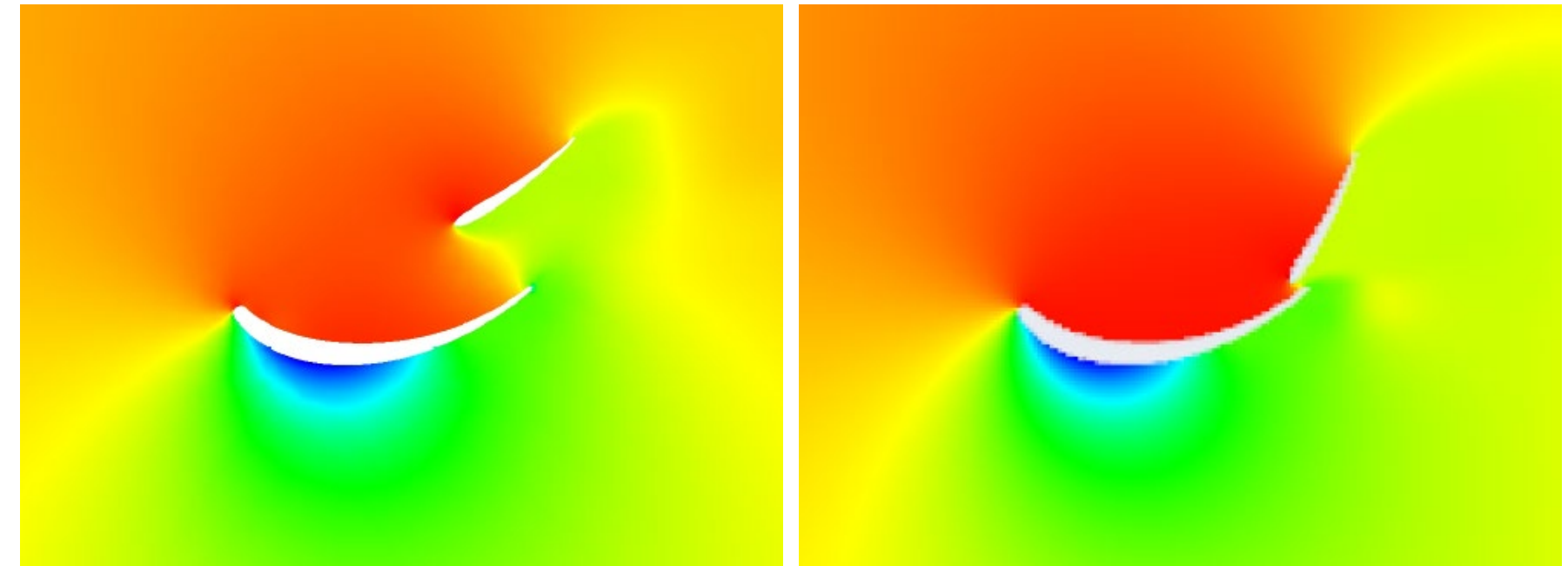
Urban pollution



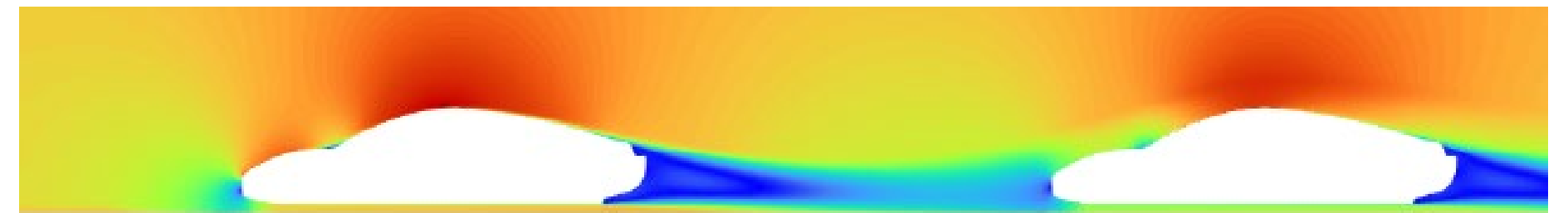
Heat exchanger



F1 rear wing

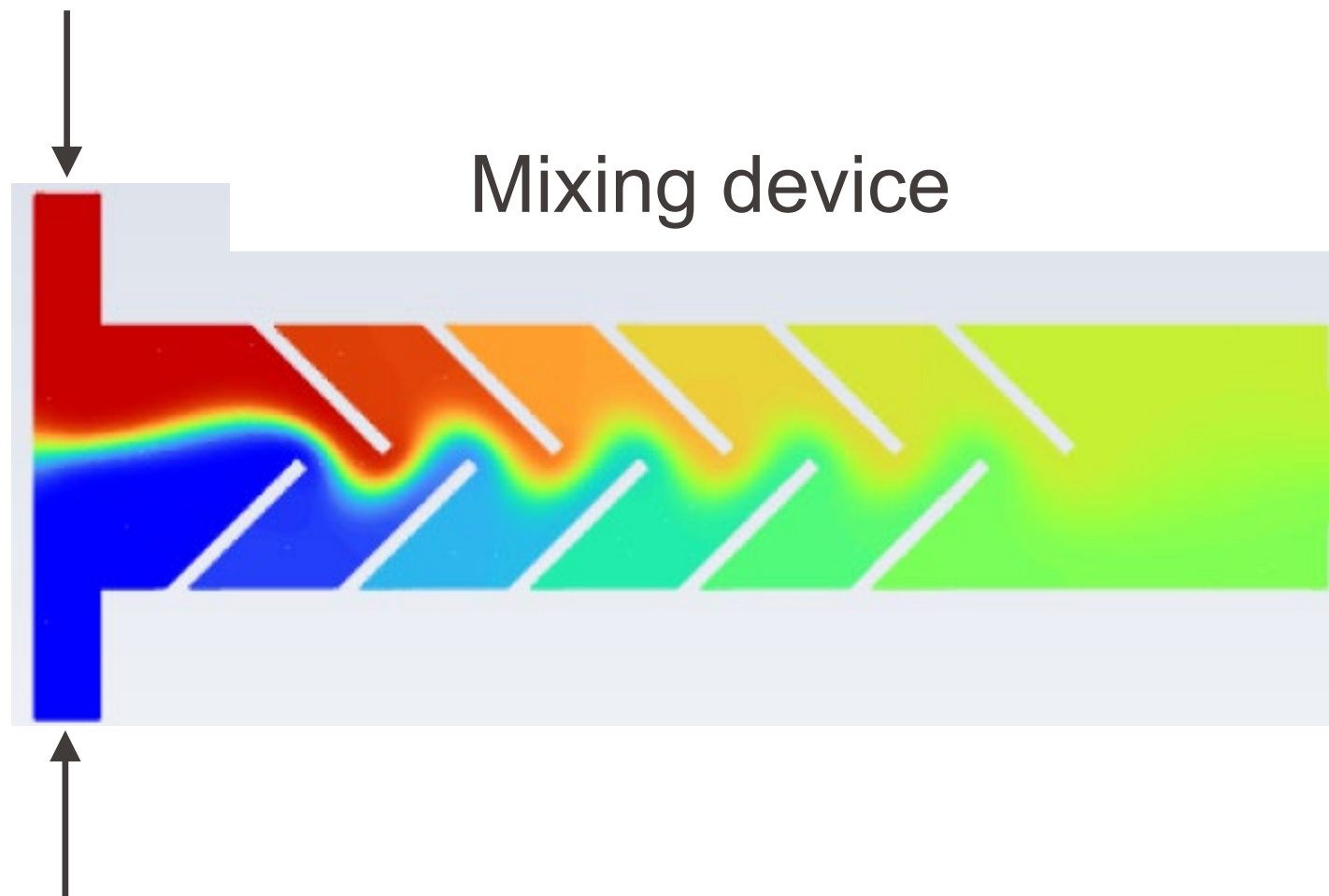


Car slipstreaming

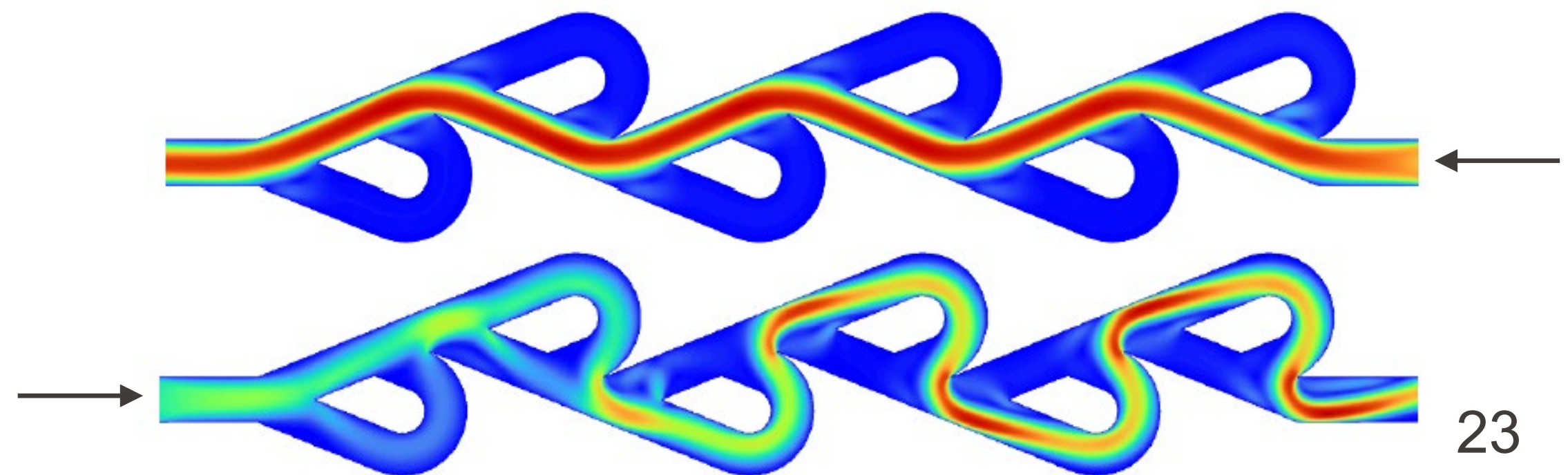


Numerical Flow Simulation

Mixing device



Tesla valve (fluidic diode)



Fluent mini-projects

- If chosen by you (may be related to other course/project, personal interest/hobby etc.), some rules to follow:
 1. Computational mesh $\leq 500'000$ elements $\rightarrow$ 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 (ME-280)
 - Numerical analysis (MATH-251)
- Recommended courses:
 - Fluid flow (ME-271)
 - Discretization methods in fluids (ME-371)
- Other concepts/skills:
 - Matlab 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