Flow case: Maglev

Numerical Flow Simulation

Edouard Boujo

Fall 2022

École
polytechnique
fédérale
de Lausanne

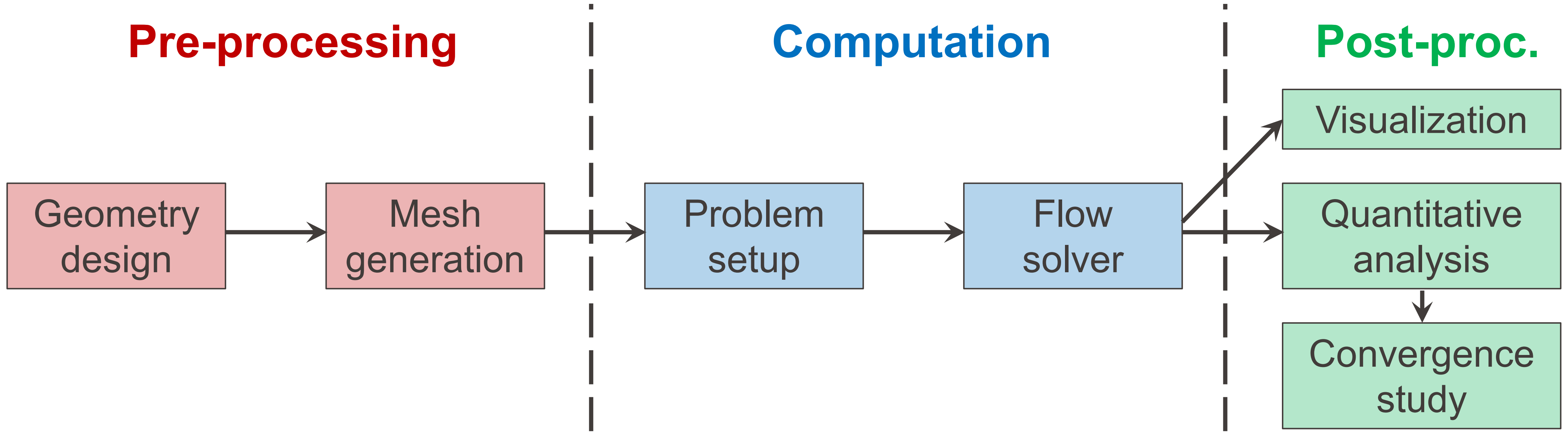
Overview

- Problem definition
  - physical problem
  - model geometry
  - flow conditions

- Numerical simulations
  - pre-processing
  - solver
  - post-processing

- Possible improvements
  - numerical simulation
  - Maglev concept

Simulation methodology: basic steps



1. Problem definition
   - define simulation goals (physical problem, important questions…)
   - identify geometrical domain

2. Pre-processing
   - create the geometry
   - create the mesh

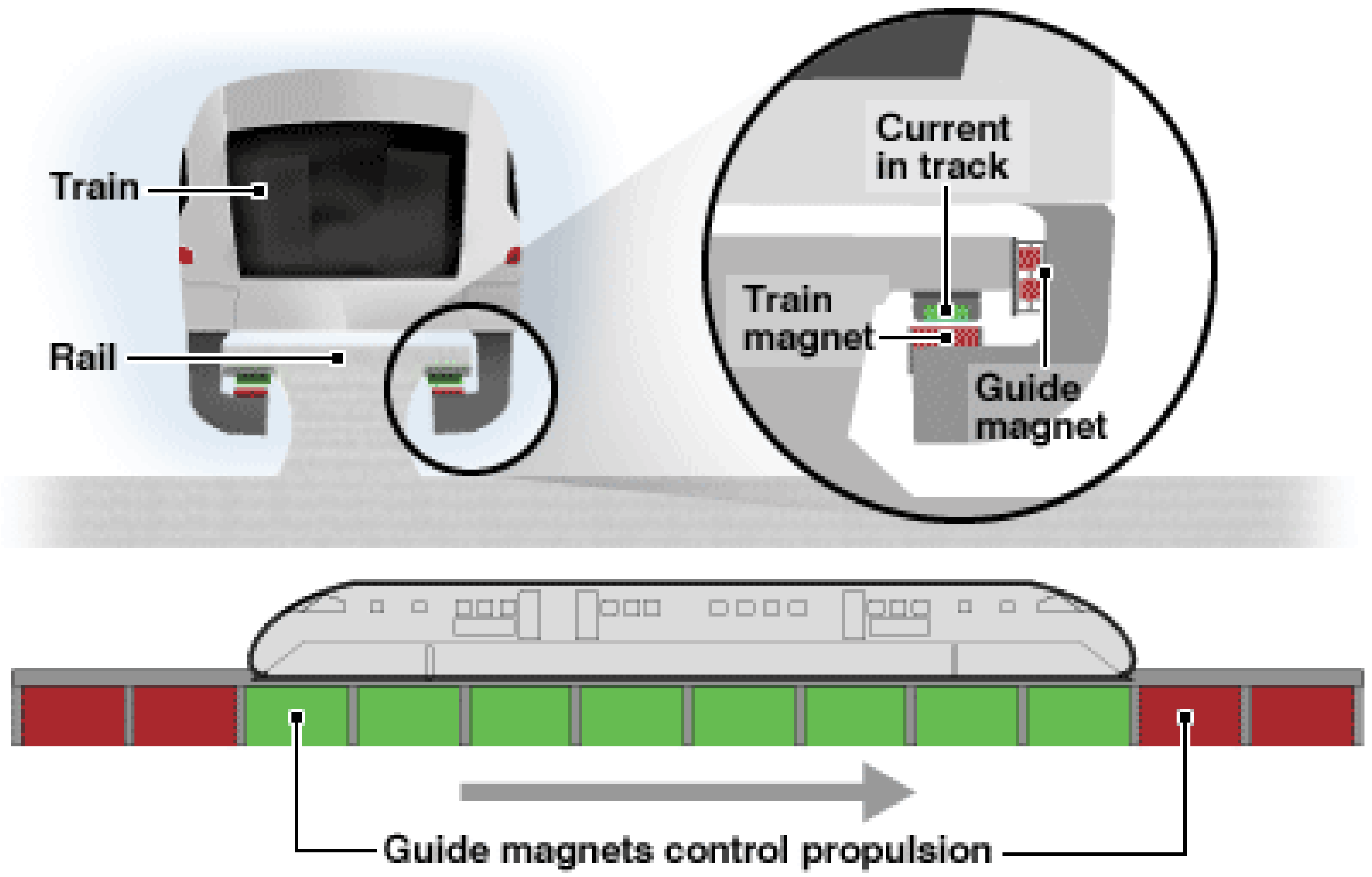
3. Computation
   - choose the physical model (turbulence, multiphase…)
   - choose the numerical method (spatial discretization, time integration…)
   - run flow solver

4. Post-processing
   - examine the numerical results (visualization, quantitative analysis)
   - extract insights regarding physical behavior

5. Improvement
   - perform verification and validation
   - revise problem definition (geometry, mesh, modeling, solver)

Physical problem

- Maglev (magnetically-levitated) vehicles for high-speed ground transport
    - magnetic levitation to provide “frictionless” interaction with the rail
    - linear motors are used for propulsion (speeds in excess of 1000 km/h)
    - aerodynamic cooling of levitation and propulsion systems



Shanghai Maglev

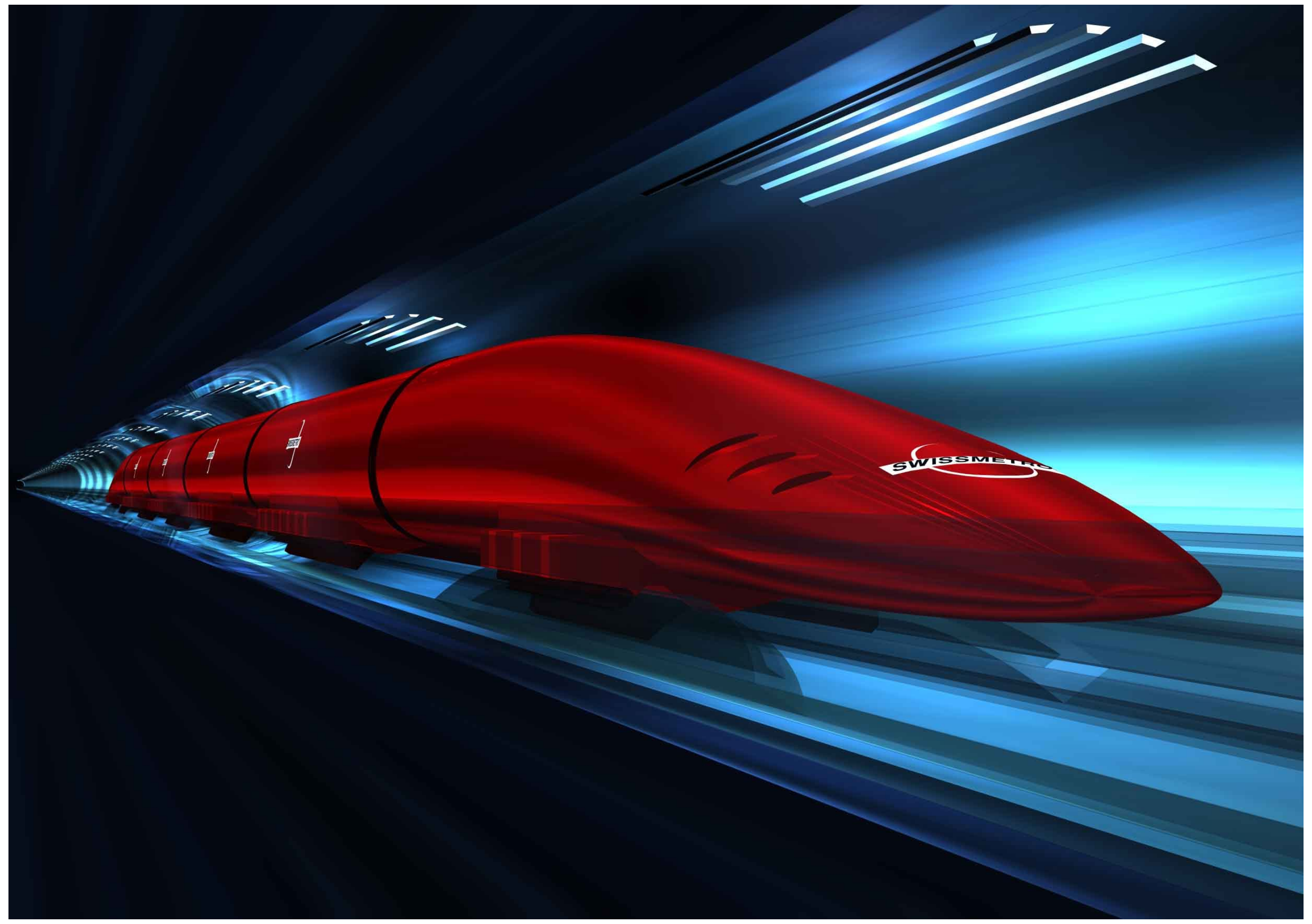
Physical problem

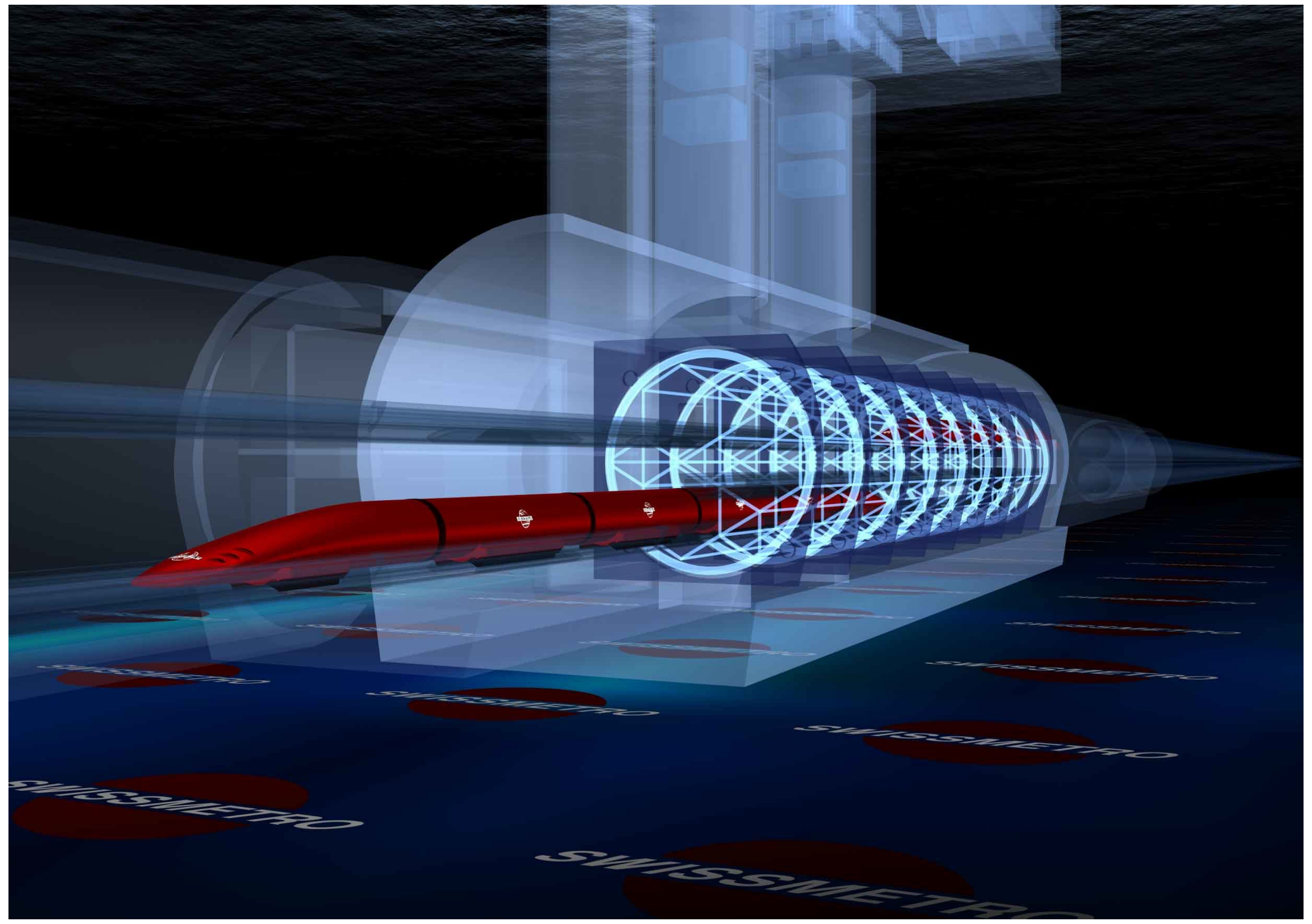
- Maglev (magnetically-levitated) vehicles for high-speed ground transport
  - SwissRapide (proposed) Maglev transportation system
  - fast (500 km/h) above-ground system
  - energy efficient, environmentally friendly, low maintenance & operation cost

www.swissrapide.com

Physical problem

- Vactrain: high-speed vehicle in a de-pressurized tunnel
  - reduced aerodynamic drag
  - underground system provides less surface disruption
  - basis for the Swissmetro concept (1974-2009)





www.swissmetro.ch

Physical problem

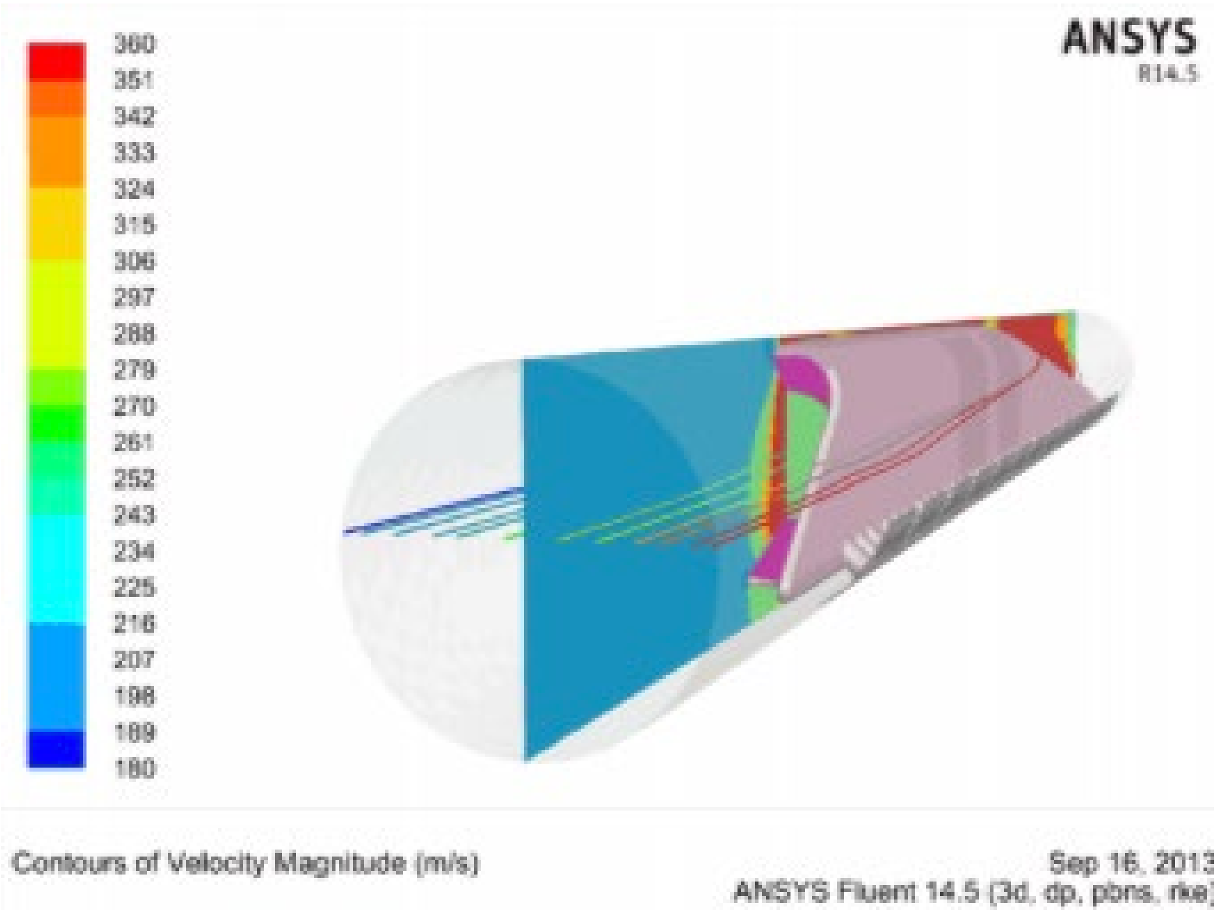
- Vactrain: high-speed vehicle in a de-pressurized tunnel
  - recently re-invented as the Hyperloop concept by Tesla and SpaceX
- Hyperloop concept
  - pod-like vehicle design, reduced pressure tubes
  - open-source approach to advance technology
  - test track has been constructed
- 2019 Hyperloop Pod Competition
  - worldwide university student competition
  - design the best self-propelled transport pod
  - sole criterion: maximum speed
  - tested in one-mile Hyperloop test track
  - EPFLoop placed 3rd

http://www.spacex.com/hyperloop

Physical problem

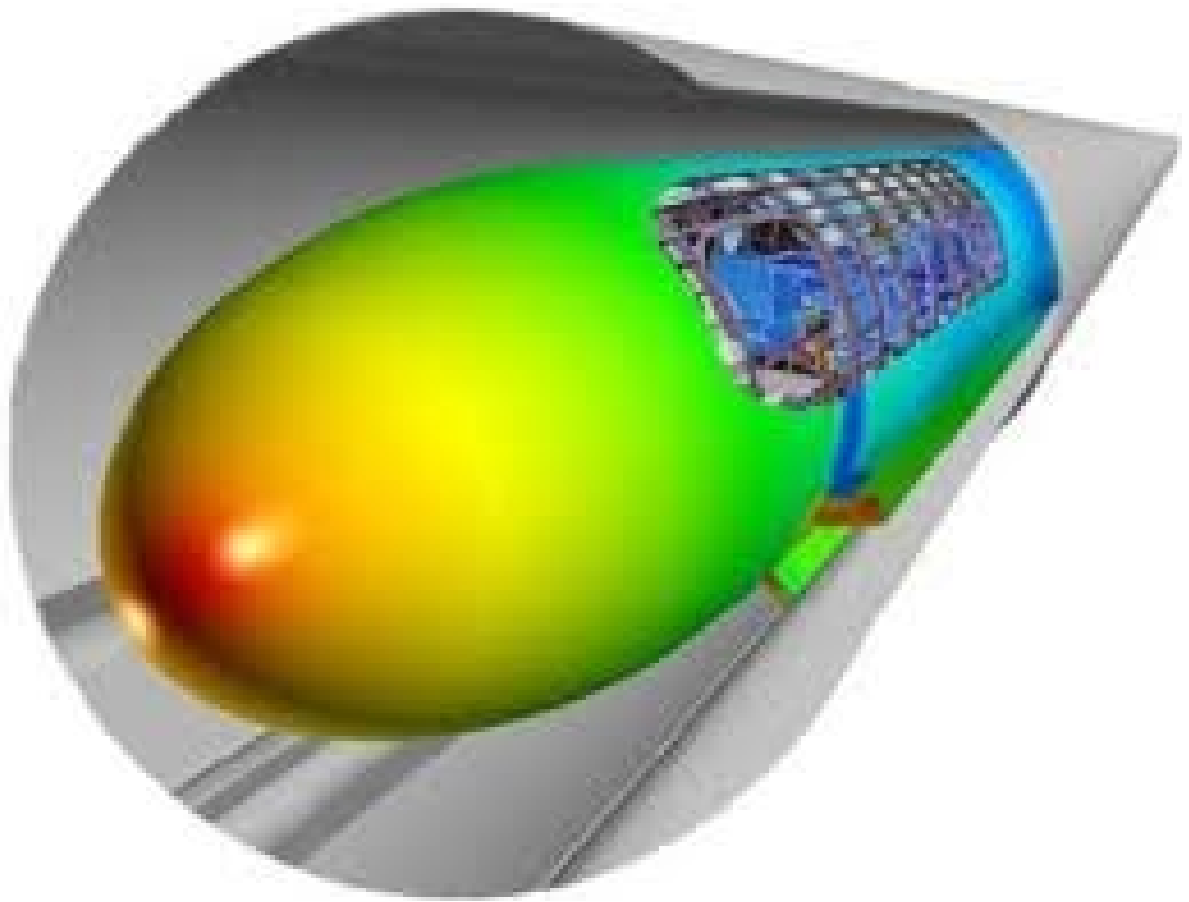
- Operation in a long tunnel
  - high blockage ratio leads to strong “piston” effect
  - aerodynamic forces different in a tunnel and in open environment
  - cooling also influenced by different aerodynamics

- In 2013, ANSYS ran CFD simulations to investigate the aerodynamics of the pod. They showed that:
  - the capsule would need to be significantly reshaped to avoid creating supersonic airflow,
  - Hyperloop has challenges but may be feasible.



Physical problem

- Goals of present numerical simulation study
  - Compute 2D solution for low-speed flow in a tunnel
    - flow fields
    - surface pressure, lift & drag forces
    - thermal effects in cooling system
  - Compute 3D solution for high-speed flow in tunnel
    - determine influence of velocity on flow & thermal properties
  - Determine influence of tunnel
    - compare solution with & without tunnel
    - optimize design of tunnel

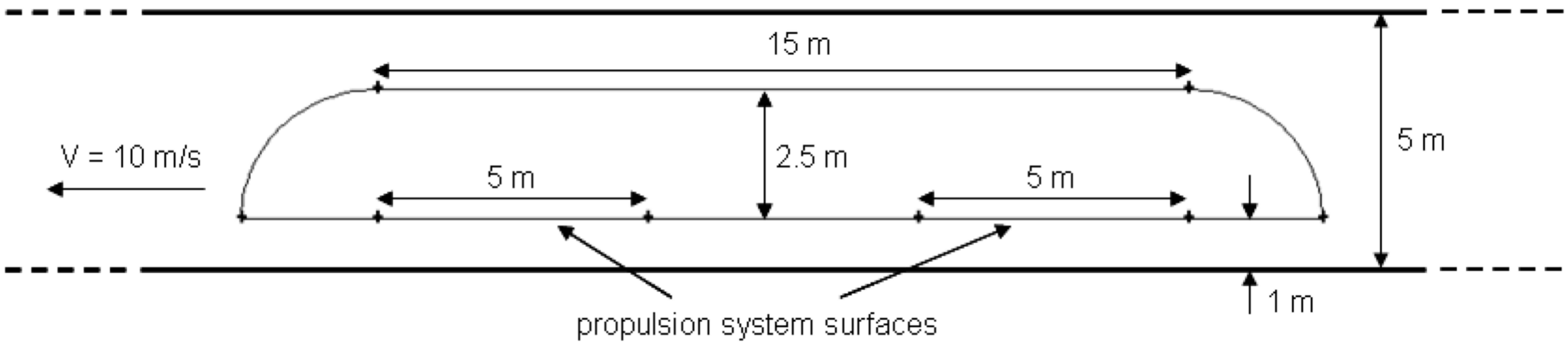


Physical problem

- Goal of this lecture: to present the procedure for NFS
  - Typical selection of choices are presented
  - These choices are not necessarily optimal; they need to be analyzed
  - “The correct solution” is not presented (since it does not exist)
  - Your goal is to propose improvements of the presented solution

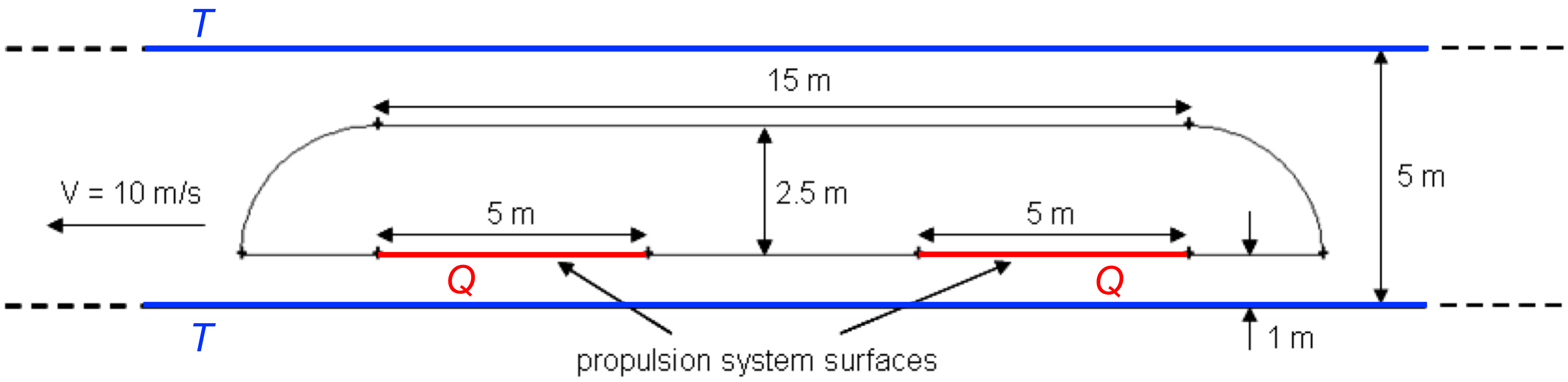
Maglev geometry

- In this study: simplified Maglev geometry
  - 2D cross-section through vertical midplane
  - Central rectangular section with circular nose and tail
  - Maglev length *L* = 20 m, Maglev diameter *d* = 2.5 m, tunnel diameter *D* = 5 m
  - Neglect central and guidance rails
  - 2 propulsion systems (electromagnets) incorporated into lower surface (investigate thermal effects)



Flow conditions

- Flow and thermal conditions:
  - vehicle speed, v = 10 m/s (36 kph)
  - constant heat flux on propulsion system surfaces, Q = 5 kW/m^2
  - all other train surfaces are assumed adiabatic
  - tunnel surface has a constant temperature, T = 15 C
- Material properties (air):
  - density, ρ = 1.225 kg/m^3
  - dynamic viscosity, μ = 1.7894x10^{-5} Pa s
  - specific heat capacity at constant pressure, C_p = 1.006.43 J/kg K
  - thermal conductivity, k = 0.0242 W/m K
  - upstream turbulence intensity = 5%



Flow conditions

- Dimensionless numbers
  - Reynolds number
    - $Re = \rho_\infty V_\infty d/\mu = 1.7x10^6 >> 1$
    - Fully turbulent flow
    - Thin boundary layer (thickness $\delta \sim x/Re^{1/2}$)
      → choice of wall treatment for turbulence model
  - Mach number
    - $M_\infty = V_\infty/c = V_\infty/(\gamma p_\infty/\rho_\infty)^{1/2} = 0.03 << 1$
    - Flow can be considered as incompressible
      → choice of physical model for fluid
      → choice of numerical method

Flow conditions

- Dimensionless numbers
  - Prandtl number (depends only on material)
    - *Pr* = momentum diffusivity / thermal diffusivity = $\mu/(k/C_p) = 0.74$
    - Thermal boundary layer, $\delta_t \sim \delta/Pr^{-1/3} \sim 1.1\delta$

      → thermal boundary layer as thin as velocity boundary layer

  - Brinkman number
    - *Br* = heat produced by viscous dissipation / heat transported by conduction

      $= \mu V^2/k(T_w - T_f) \ll 1$ for air in general

      → viscous heating effects can be neglected (in turbulence model)

  - Nusselt number
    - *Nu* = convective heat transfer / conductive heat transfer

      $= h/(k/L)$ (with $h = Q/\Delta T$ the heat transfer coefficient) $> 10^3 \gg 1$

      → heat produced will be transported, not conducted

Flow conditions

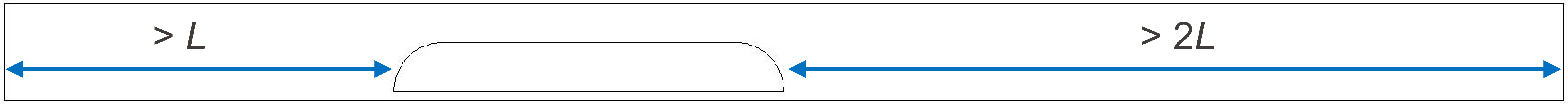
- Other geometrical / physical considerations

  - Boundary layers: decide which boundary layers are important

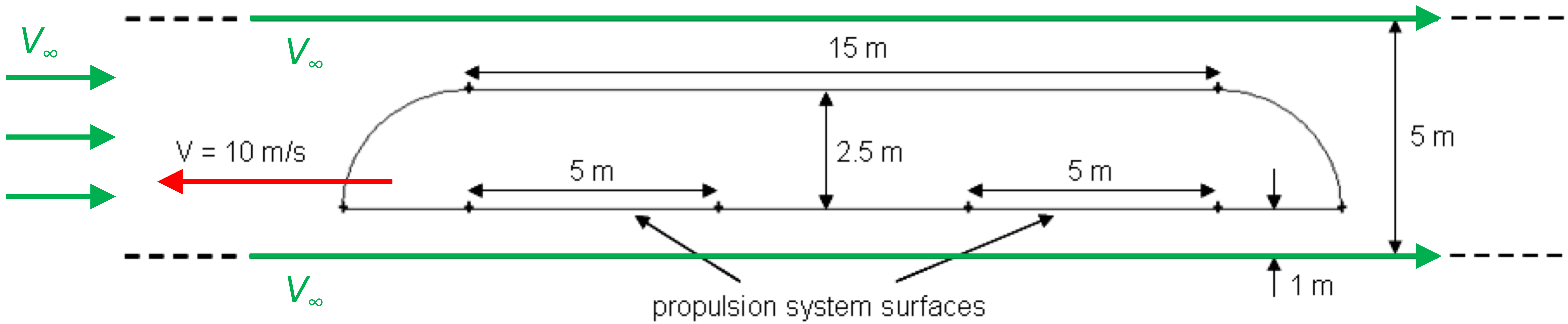
  - Bluff body: possible separation in tail region → need suitable turbulence model to capture accurately separation point.

Pre-processing

- Choose locations of inlet and outlet



- Change of frame of reference
  - Use frame in which Maglev is stationary
  - $V_\infty = -V_{Maglev}$
  - Note: tunnel is moving in this frame of reference!



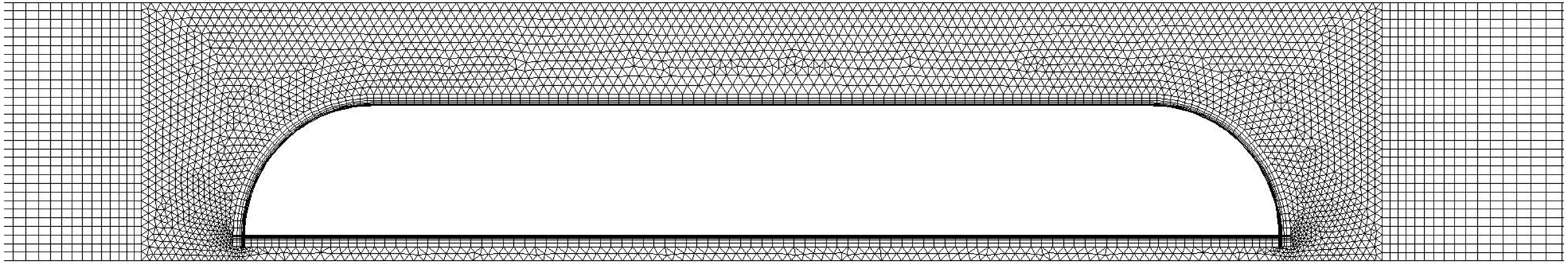
Pre-processing

- Mesh
  - Specific physics to capture:
    - boundary layers (Maglev & tunnel walls)
    - separation point, wake
  - Type of mesh?
    - structured / unstructured / block-structured / hybrid
    - triangular / quadrilateral / mixed

Pre-processing

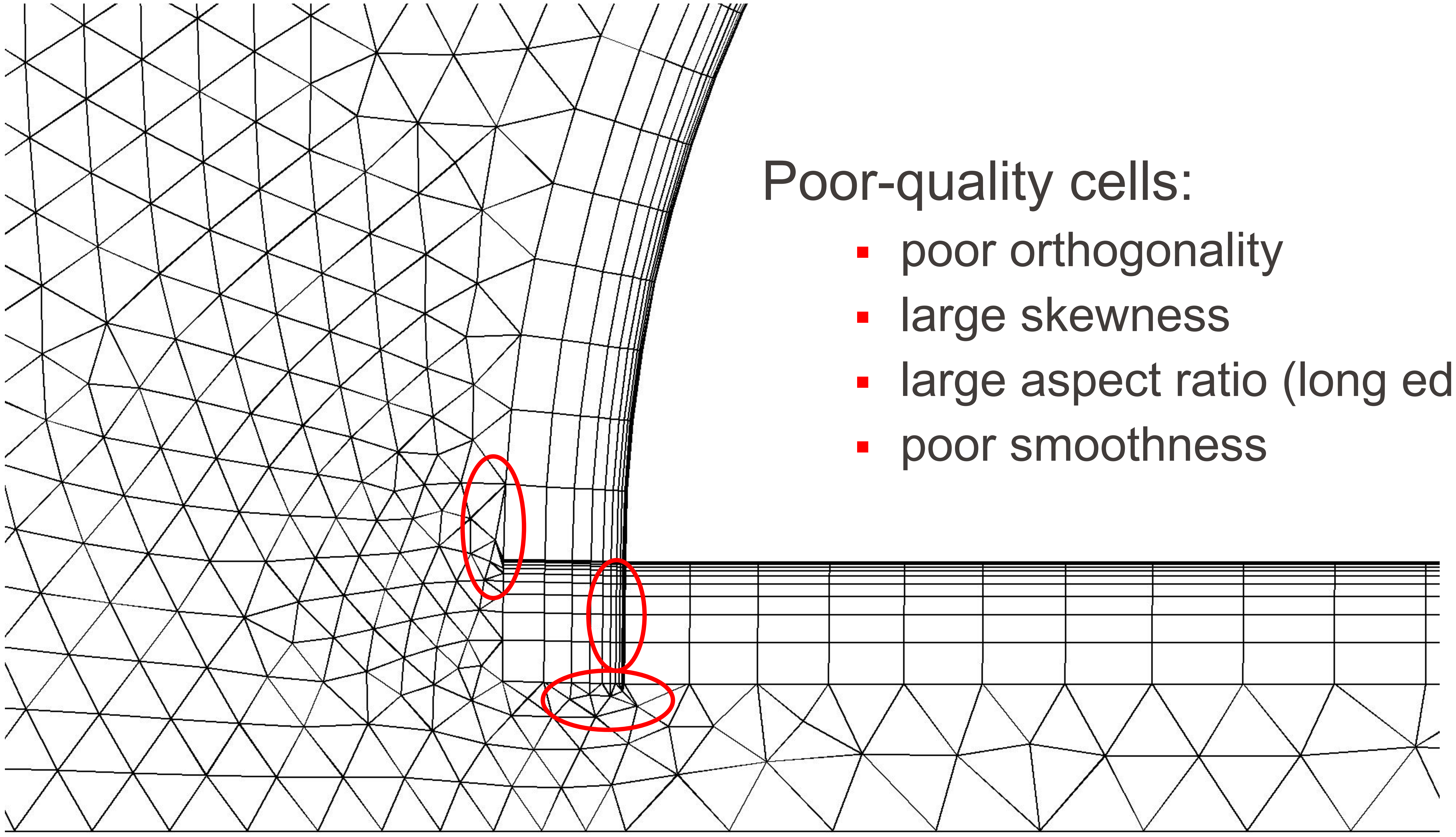
- Mesh
  - Create a mesh that is not too coarse / too fine
  - Neglect boundary layers on tunnel walls

- Mesh properties
  - hybrid: structured (quad) in boundary layer and tunnel ends + unstructured (tri) (other suitable mesh types exist)
  - 11’220 cells
  - minimum orthogonal quality well below 0.01 → some poor-quality cells!



Pre-processing

- Mesh (close-up): should resolve problems in tail / nose regions



Poor-quality cells:

- poor orthogonality
- large skewness
- large aspect ratio (long edge not in the flow direction)
- poor smoothness

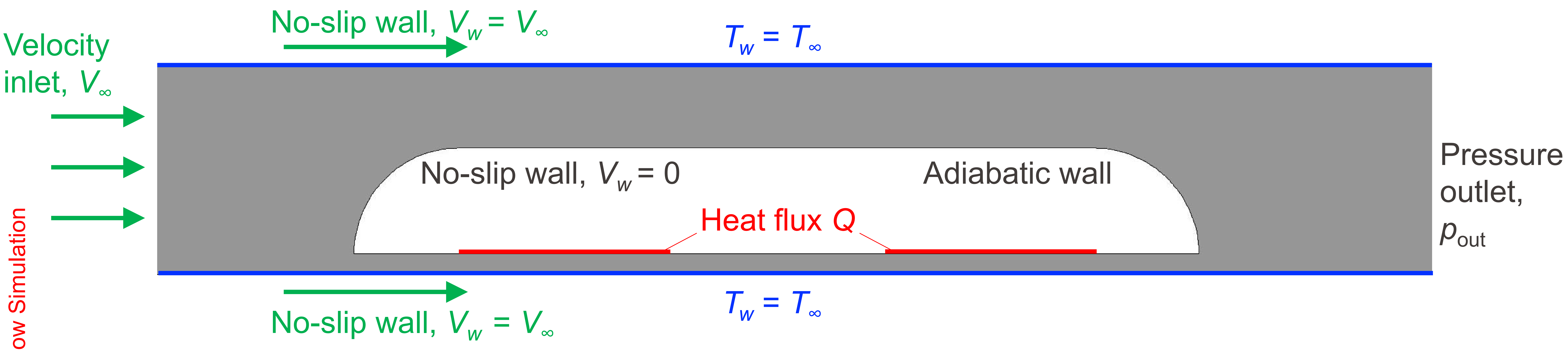
Solver set-up

- Physical models
  - Ideal gas law, default air properties (atmospheric conditions)
  - *k*-ε turbulence model (*k*-ω also appropriate)
    - RNG (“realizable” also possible; “standard” doesn’t capture accurately separation)
    - Near-wall treatment: non-equilibrium wall functions (for complex flows, e.g. separation)
    - Neglect viscous heating (Brinkman number $Br \sim 10^{-3} \ll 1$)

- Numerical methods
  - Steady
  - Pressure-based solver
    - SIMPLE scheme
    - Both first-order and second-order discretization have been considered
    - Default under-relaxation factors
  - Convergence: monitor temperature on propulsion system + drag / lift coeffs.

Solver set-up

- Boundary conditions (flow + thermal)



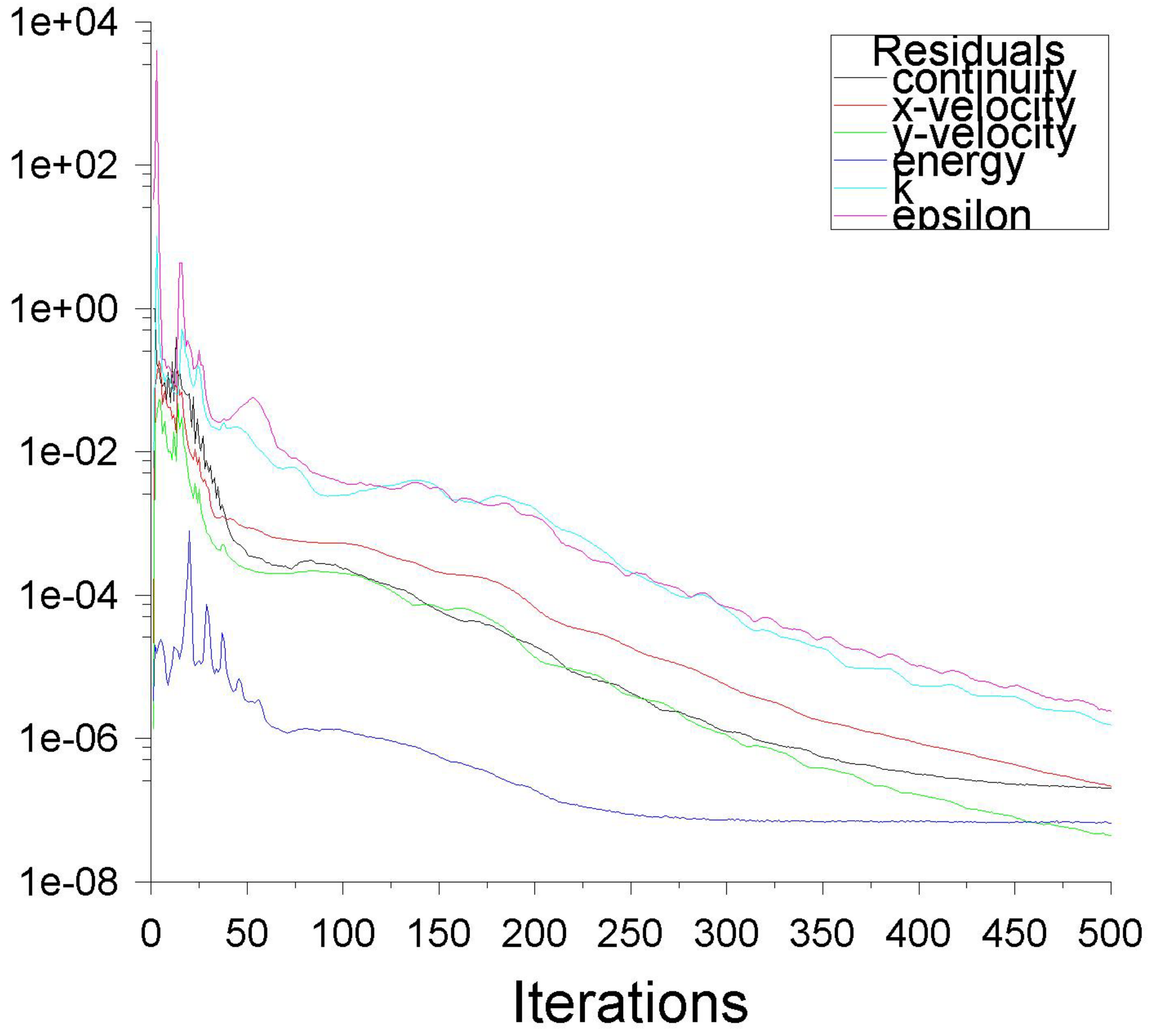
Numerical Flow Simulation

Solver set-up

- Reference values
  - Needed for pressure coefficient $C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty V_\infty^2}$, drag / lift coeffs $C_{D/L} = \frac{F_{D/L}}{\frac{1}{2}\rho_\infty V_\infty^2 A}$
  - 2D case = 3D with a depth of 1 m
  - Length: Maglev length *L* or height *d*
  - Area: length*depth = *L**1 (typical for airfoils) or *d**1 (typical for cars)
  - Density: freestream density ρ_∞

Solver procedure

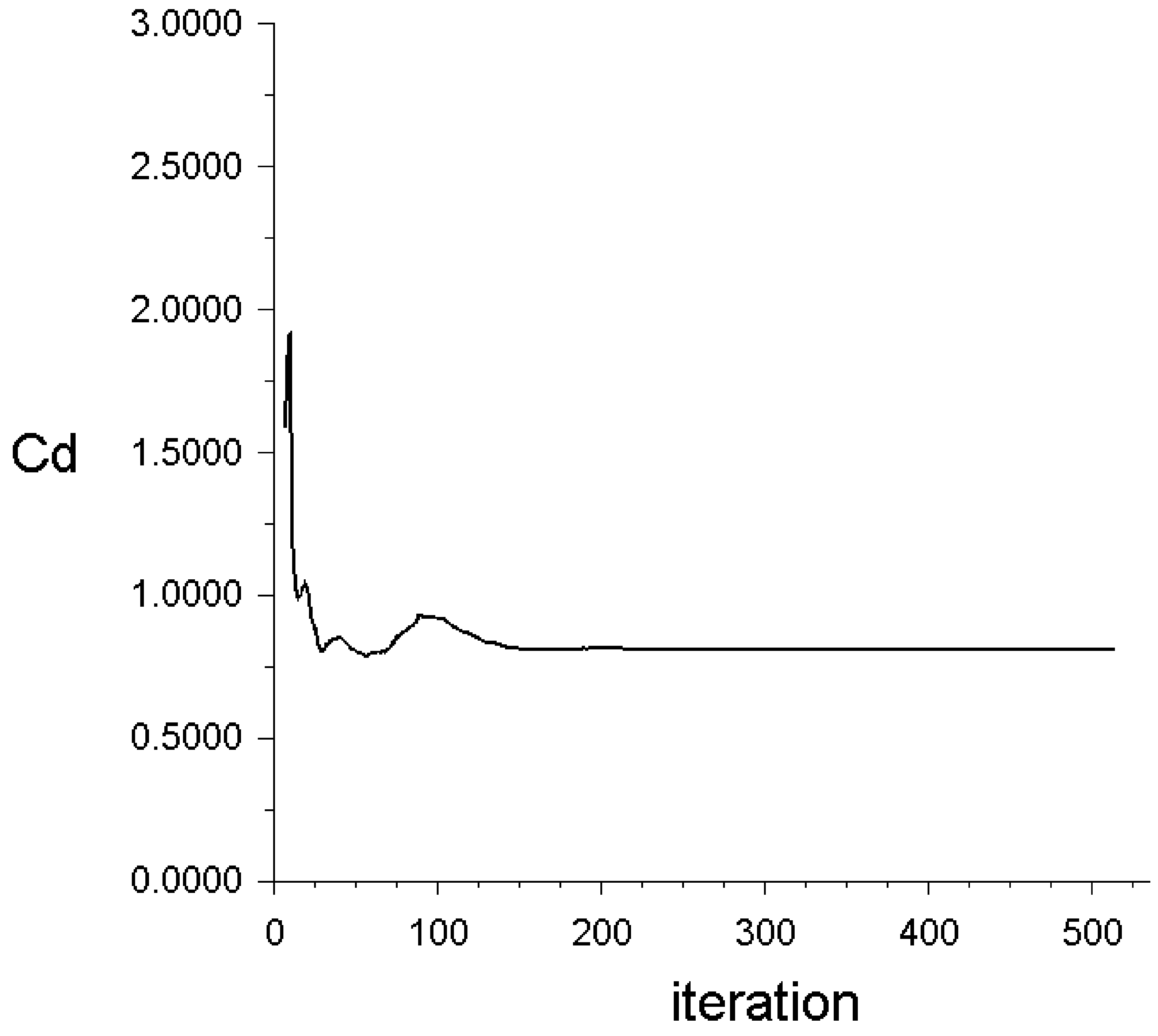
- Convergence: residuals (first-order discretization)

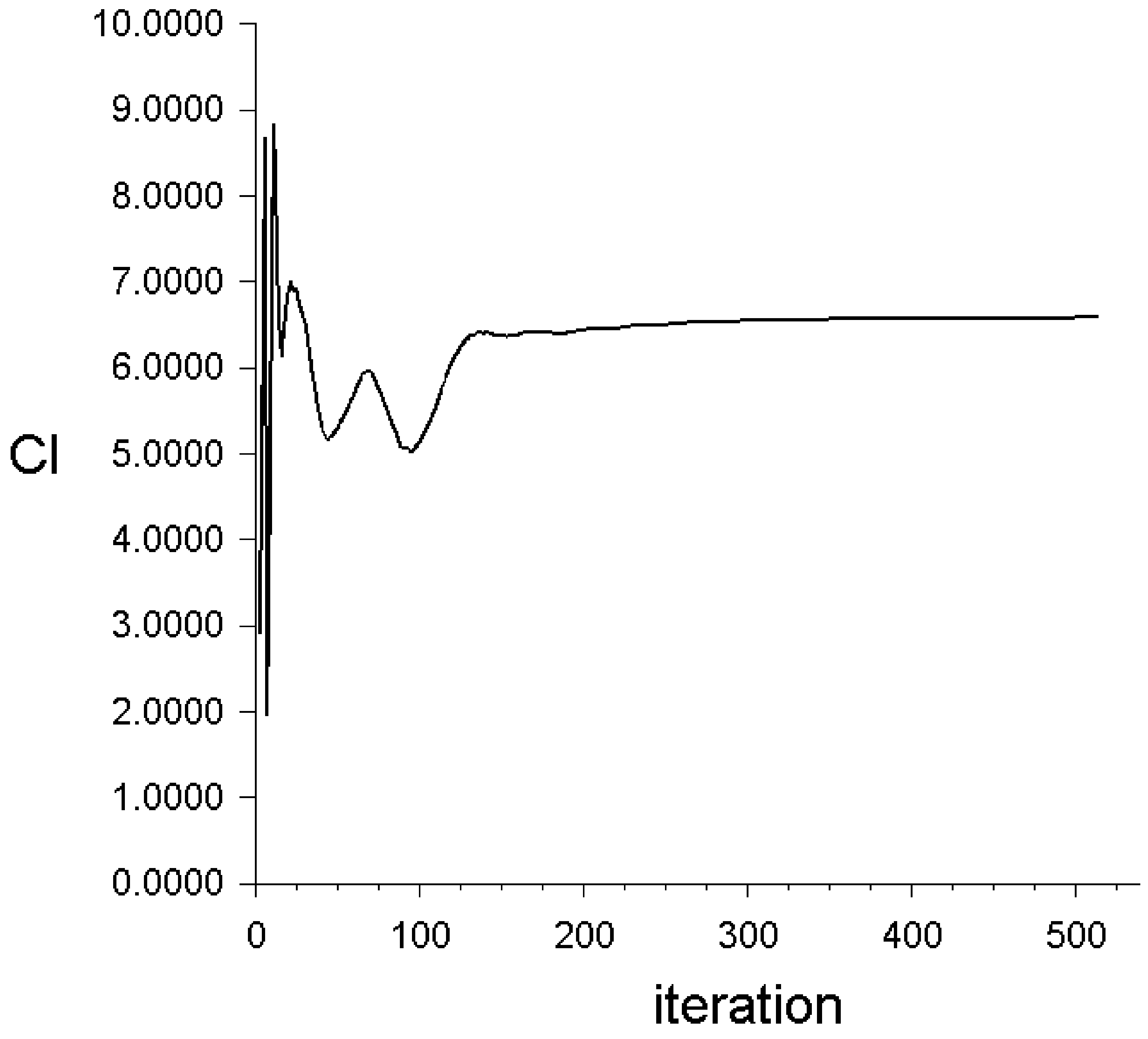


Almost all residuals below 10^{-6} after 500 iterations

Solver procedure

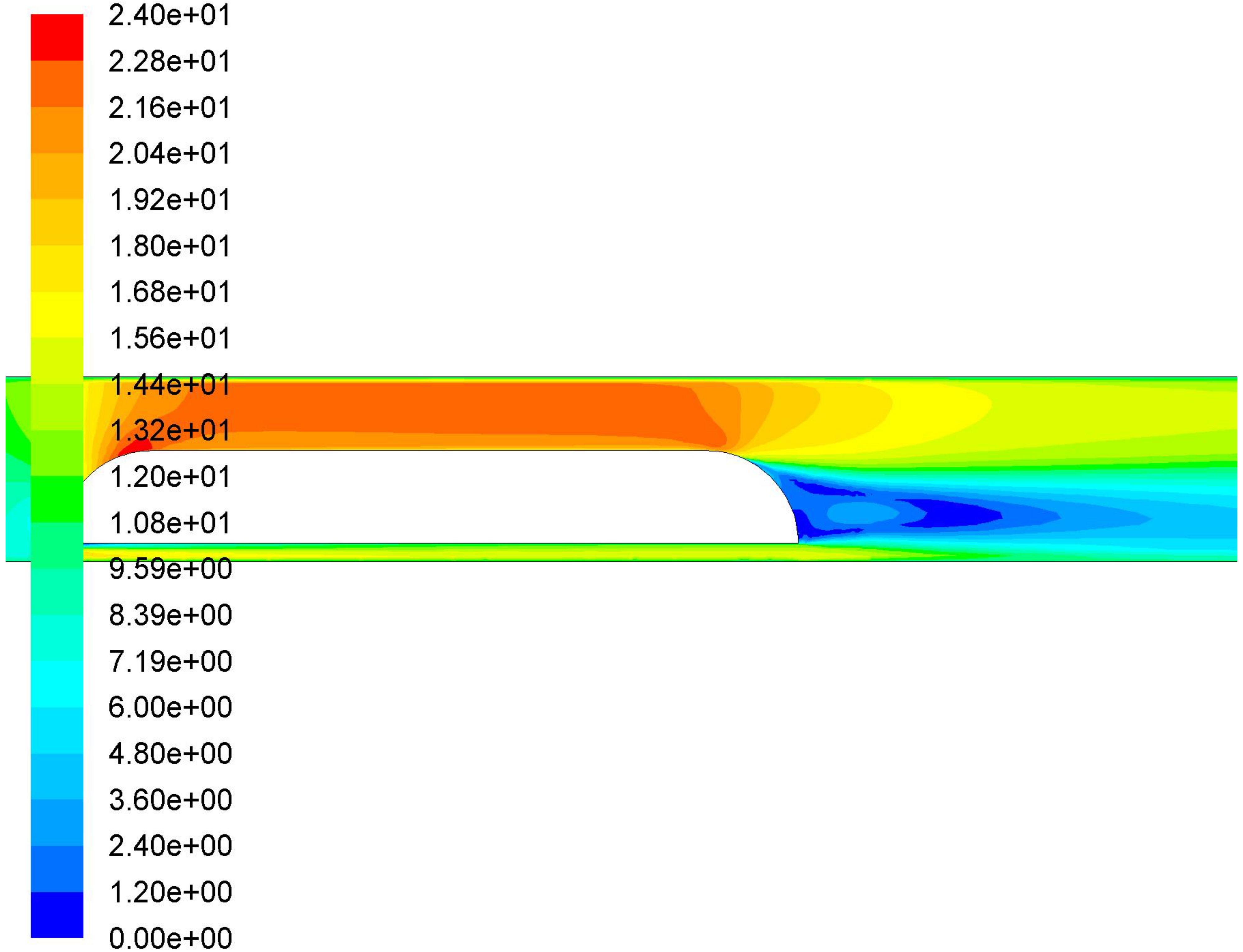
- Convergence: drag / lift coefficients





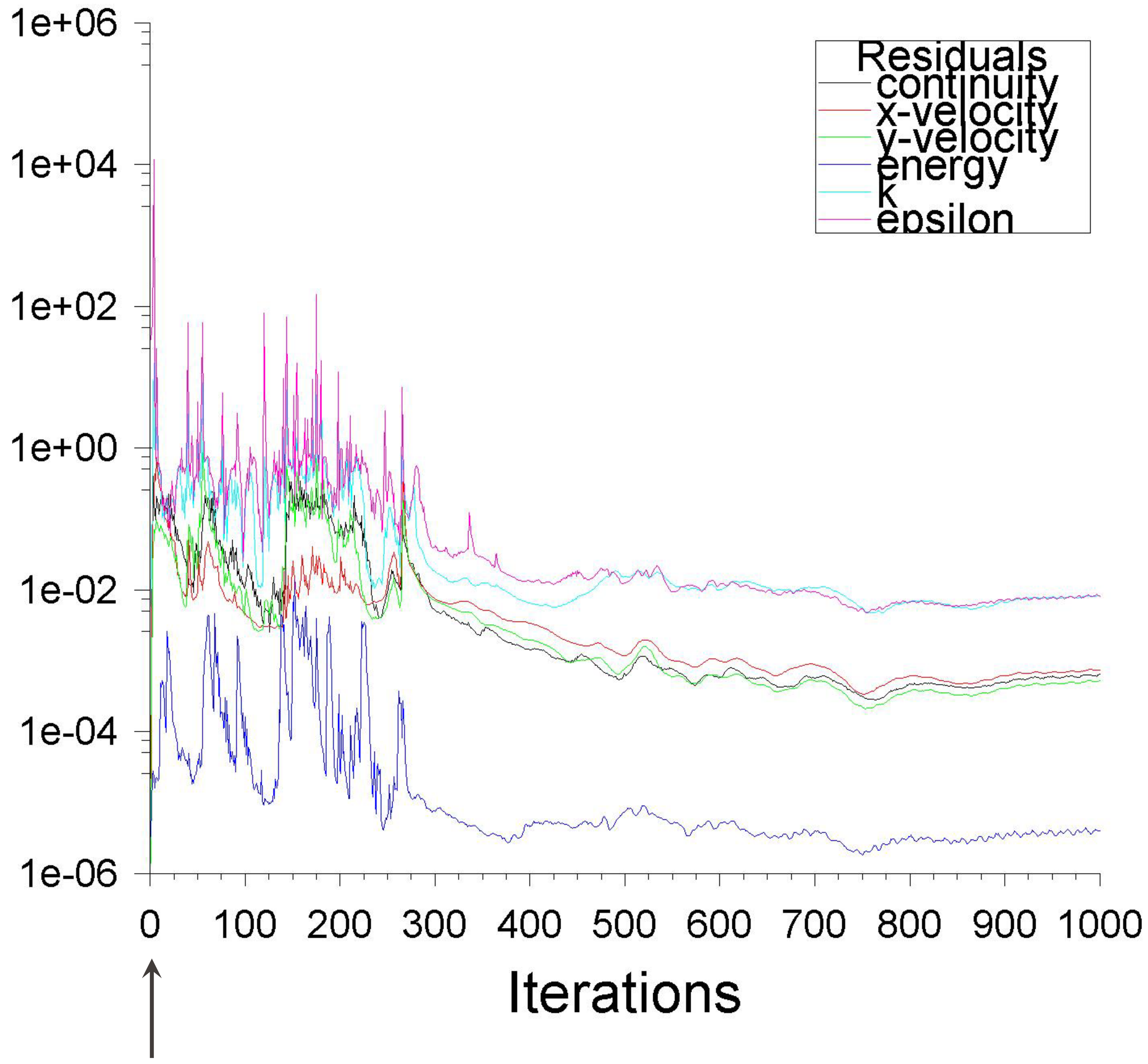
Post-processing

- Contours of velocity magnitude



Solver procedure

- Convergence: residuals (second-order discretization)

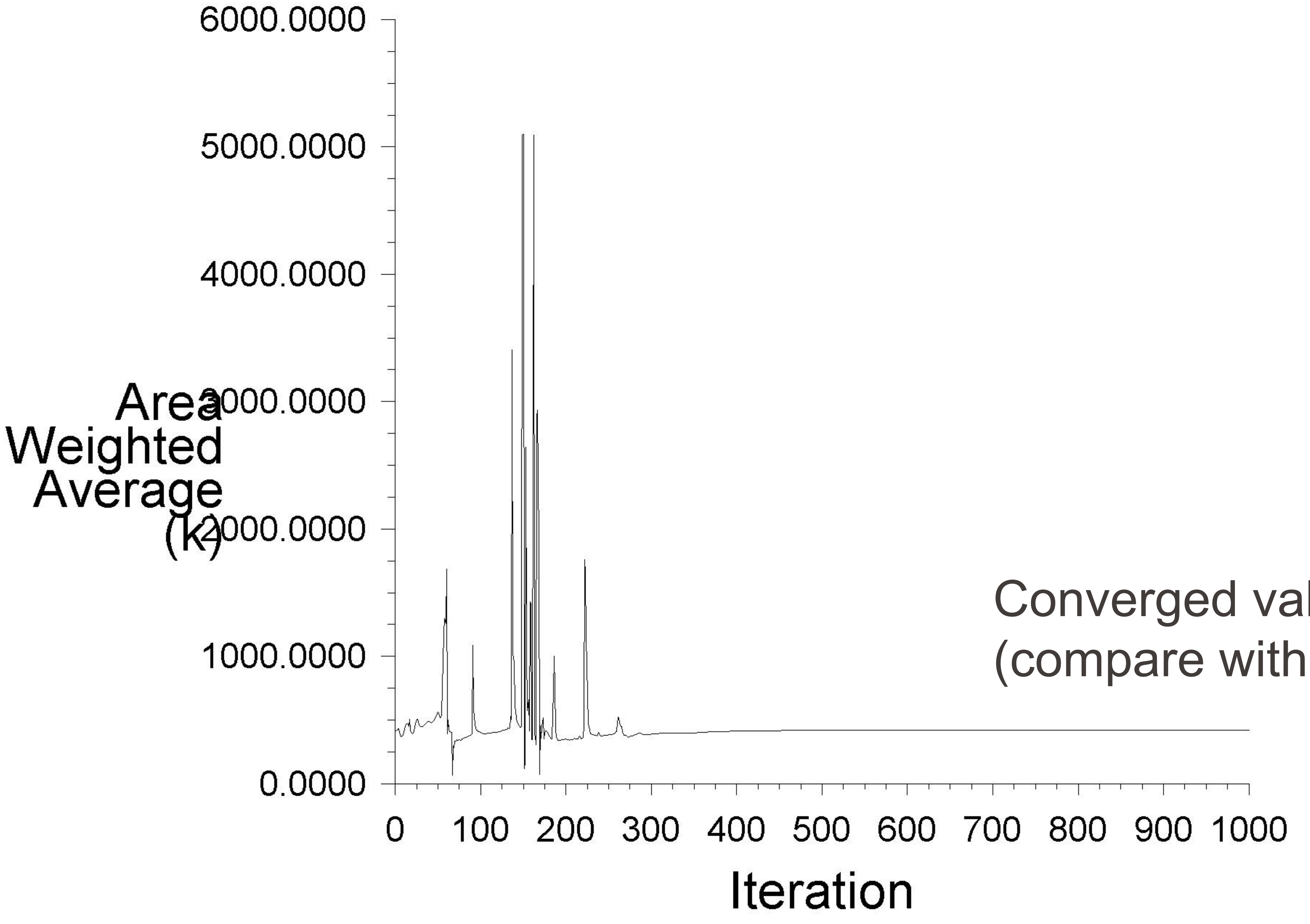


Not quite converged after 1000 iterations. The flow may actually be unsteady.

Note: reinitialized the solution; would have been smarter to restart from 1st-order solution.

Solver procedure

- Convergence: average temperature on propulsion system



Converged value: T = 419 K = 146°C
(compare with $T_{in} = T_{\infty}$ = 288 K = 15°C)

Solver procedure

- Convergence: mass conservation
  - Require net mass flux between inlet and outlet to be conserved to < 0.2 %

Mass flow rate [kg/s]	
Inlet	61.25000
Outlet	-61.23362
Net imbalance	**0.01638 (0.03 %)**

  - Mass conservation satisfied for this solution.

Solver procedure

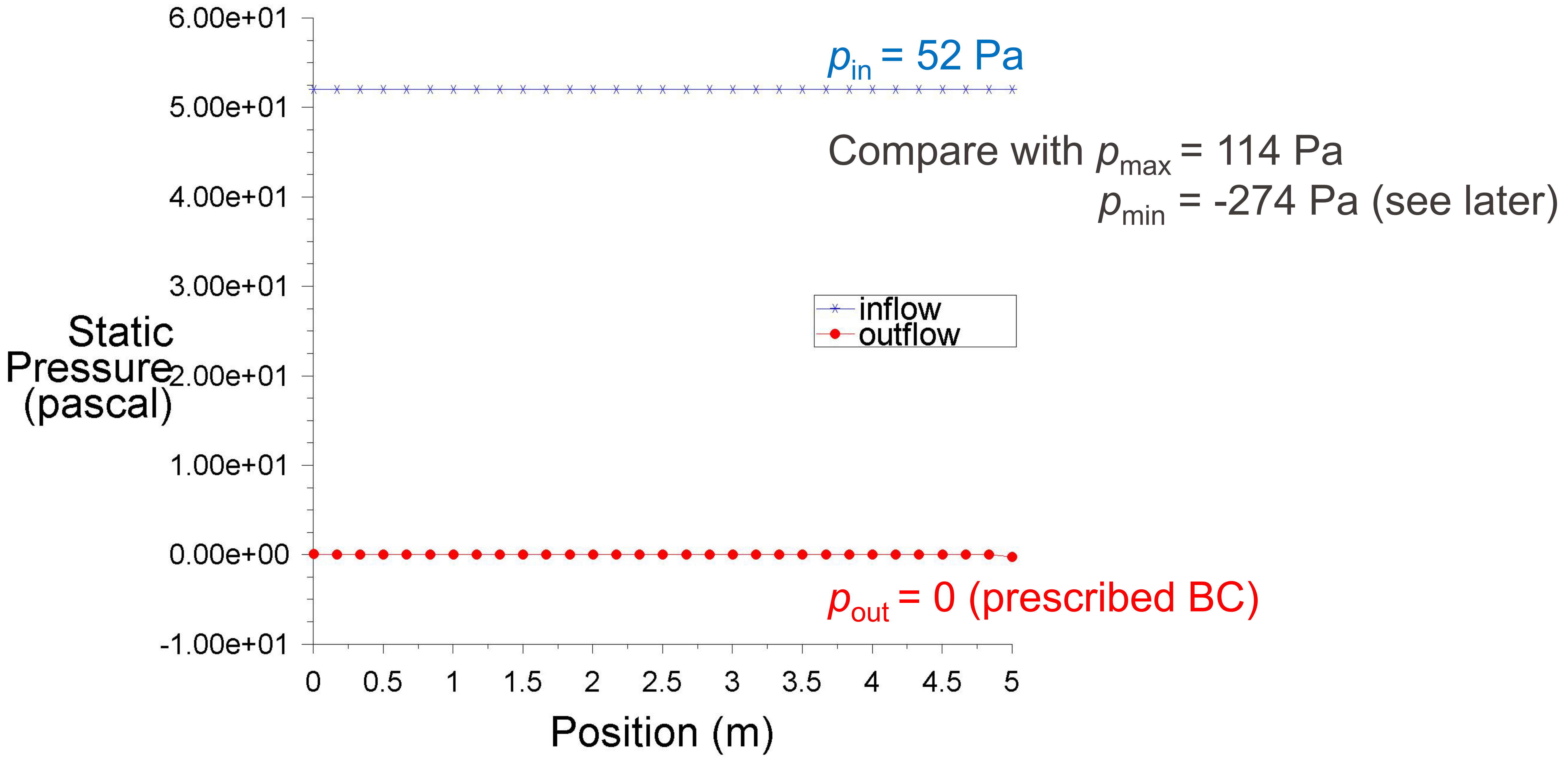
- Convergence: energy conservation
  - Require net heat flux between inlet and outlet to be conserved to < 0.2 %

Heat flow rate [kW]	
Inlet	-615.82
Propulsion	50
Maglev	0
Tunnel	1.48
Outlet	566.84
Net imbalance	**-0.46 (0.07 %)**

  - Energy conservation satisfied for this solution.

Post-processing

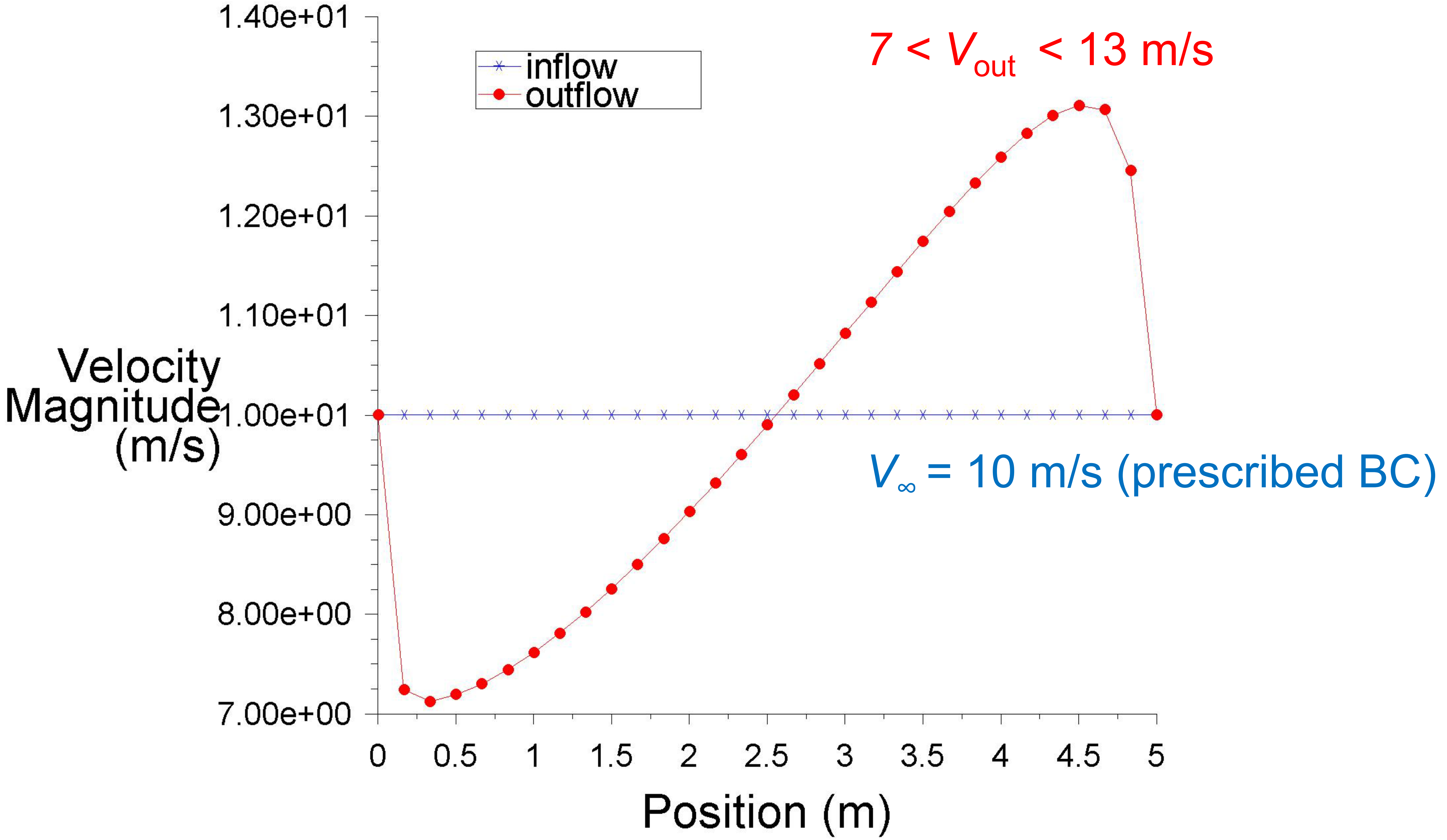
- Control: inlet / outlet pressure



Post-processing

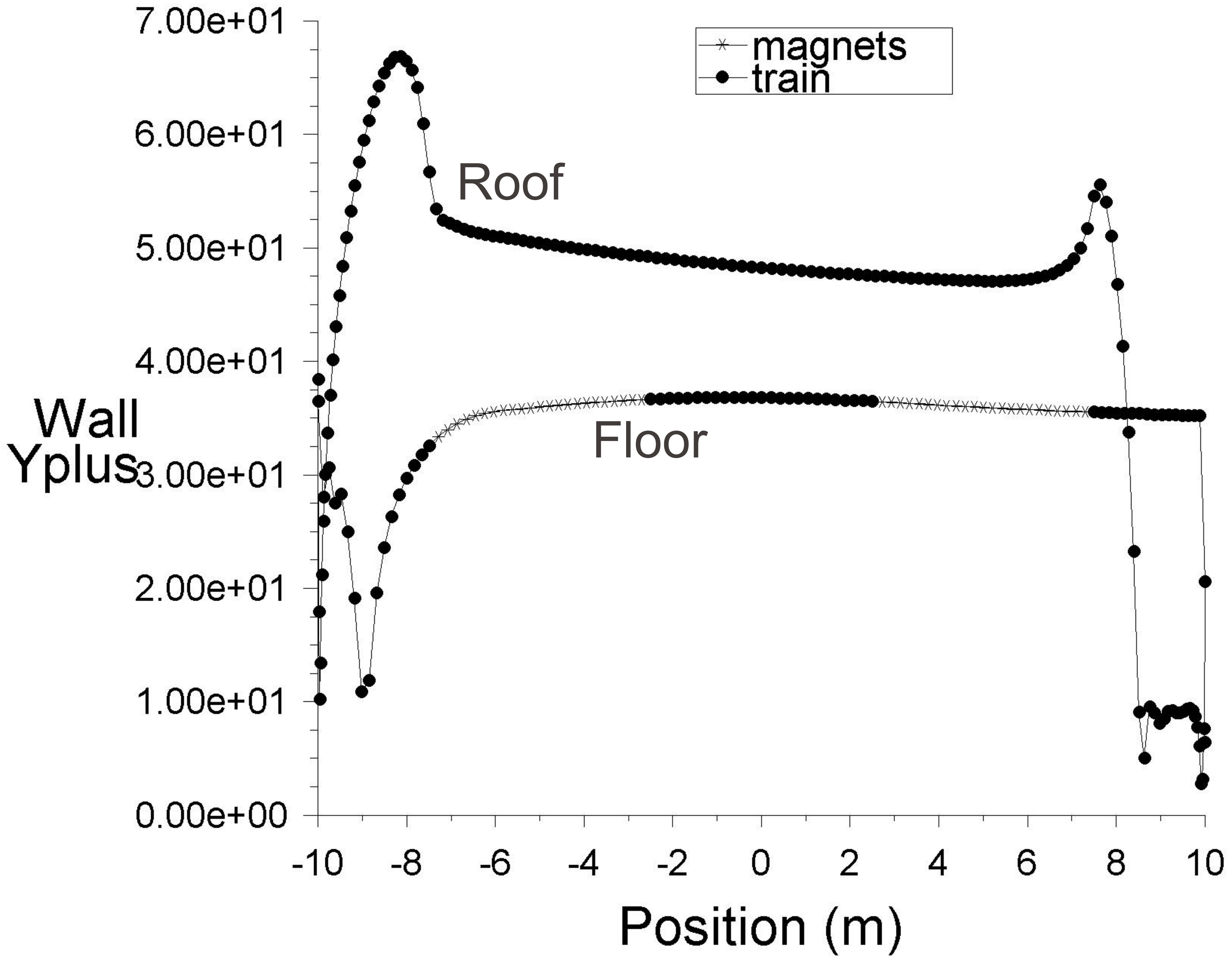
- Control: inlet / inlet velocity

Compare with V_{max} = 24 m/s (see later)



Post-processing

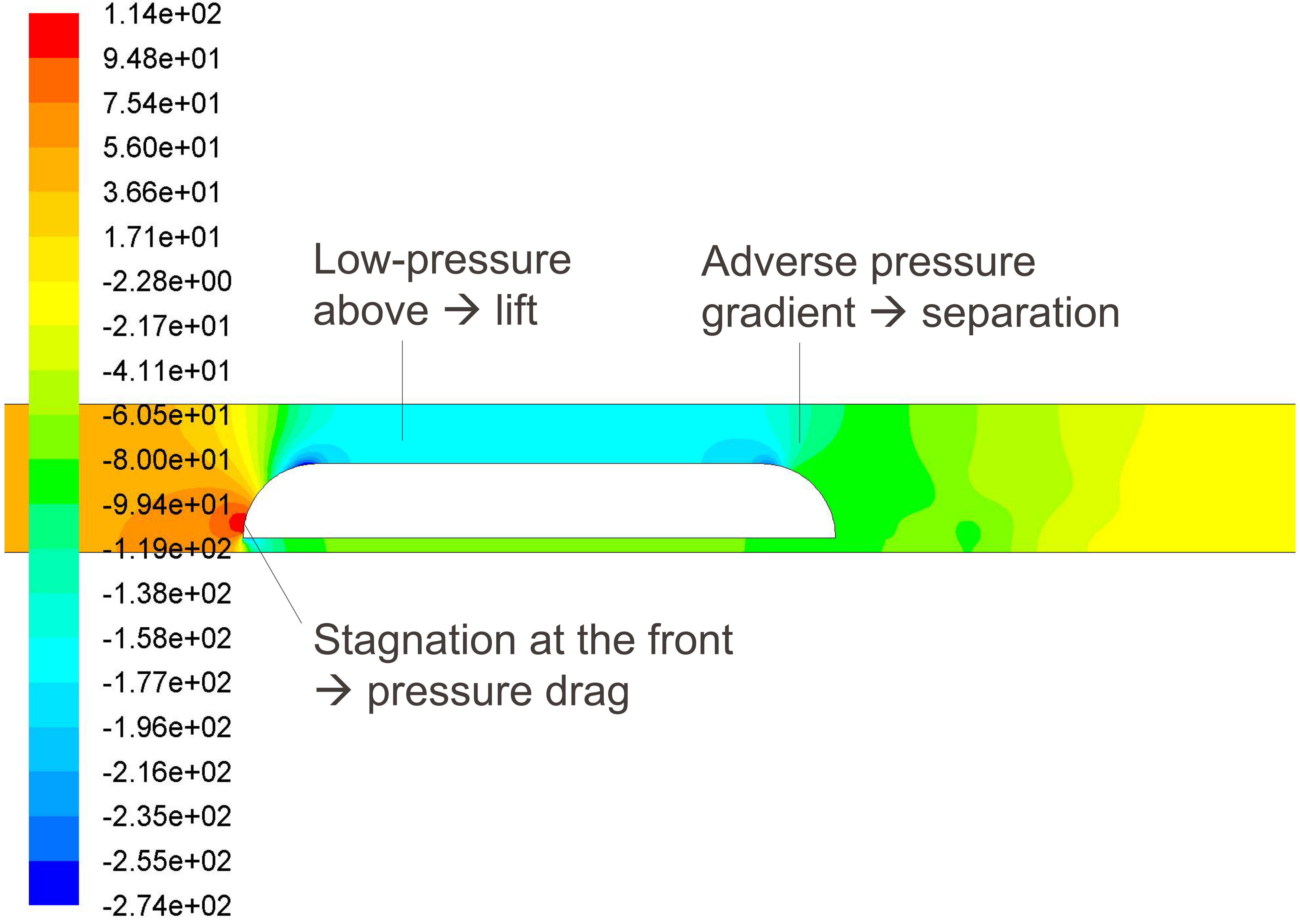
- Control: first cell value of y^+ along Maglev wall



Numerical Flow Simulation

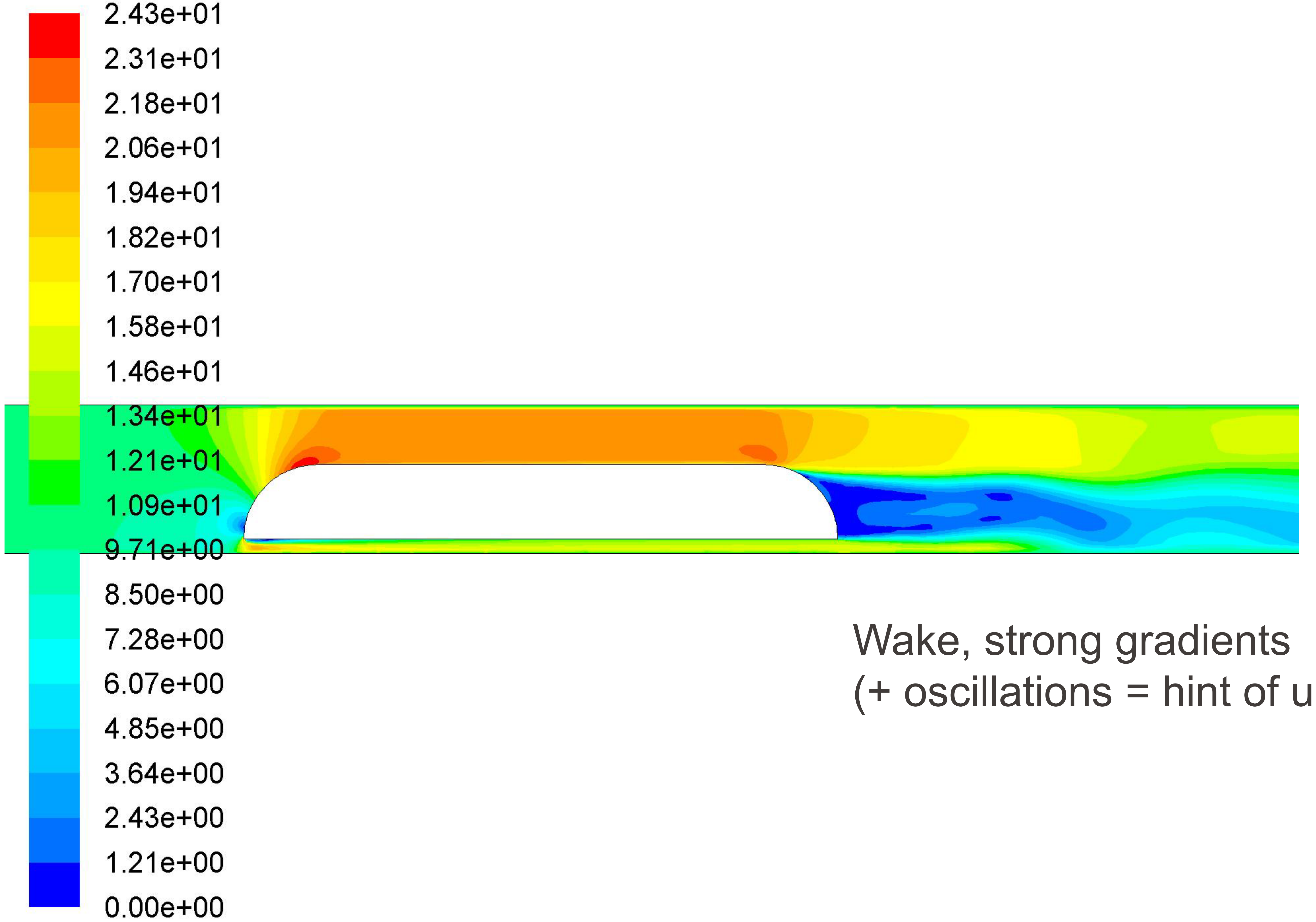
Post-processing

- Contours of pressure



Post-processing

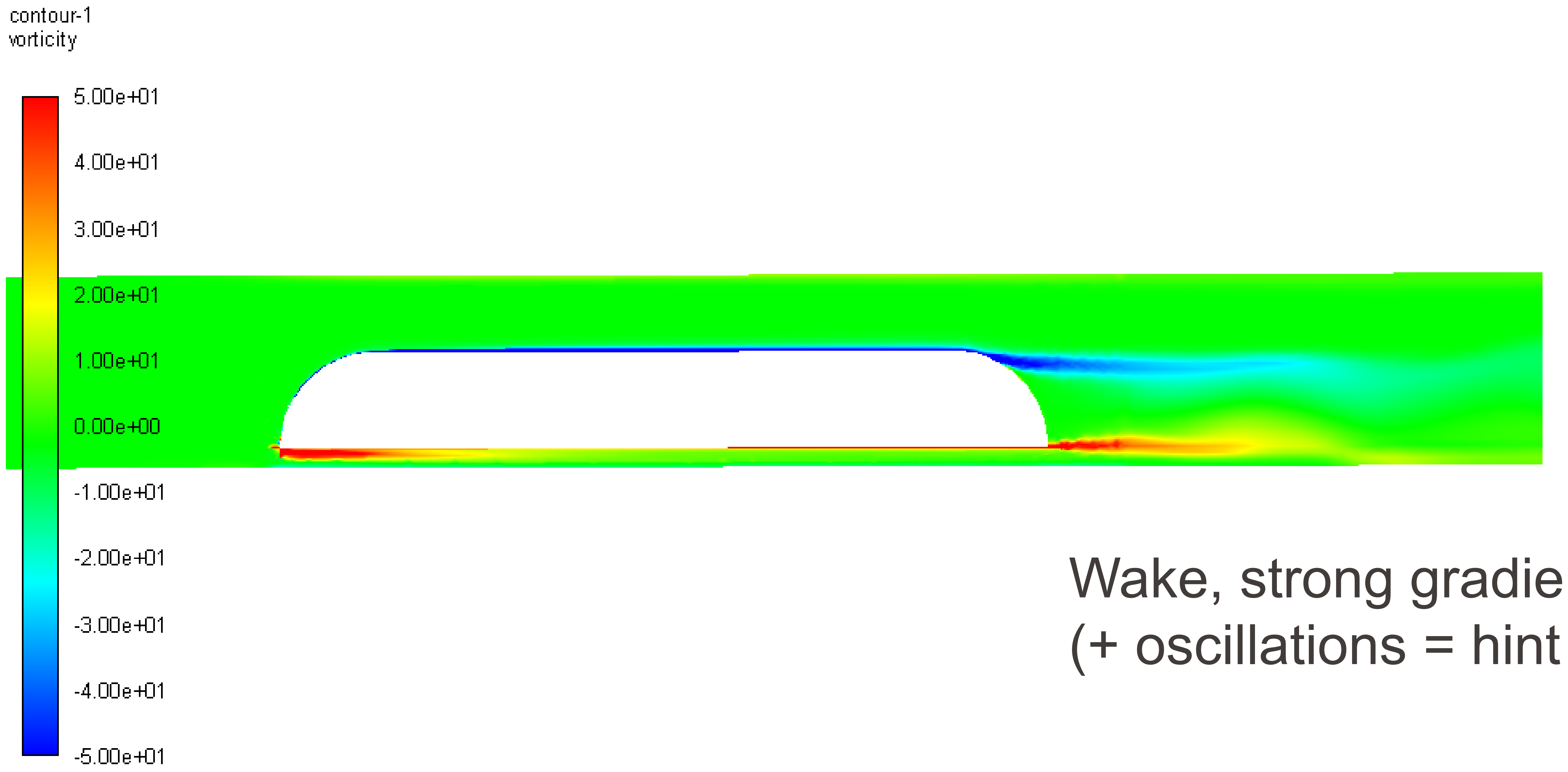
- Contours of velocity magnitude



Wake, strong gradients
(+ oscillations = hint of unsteadiness)

Post-processing

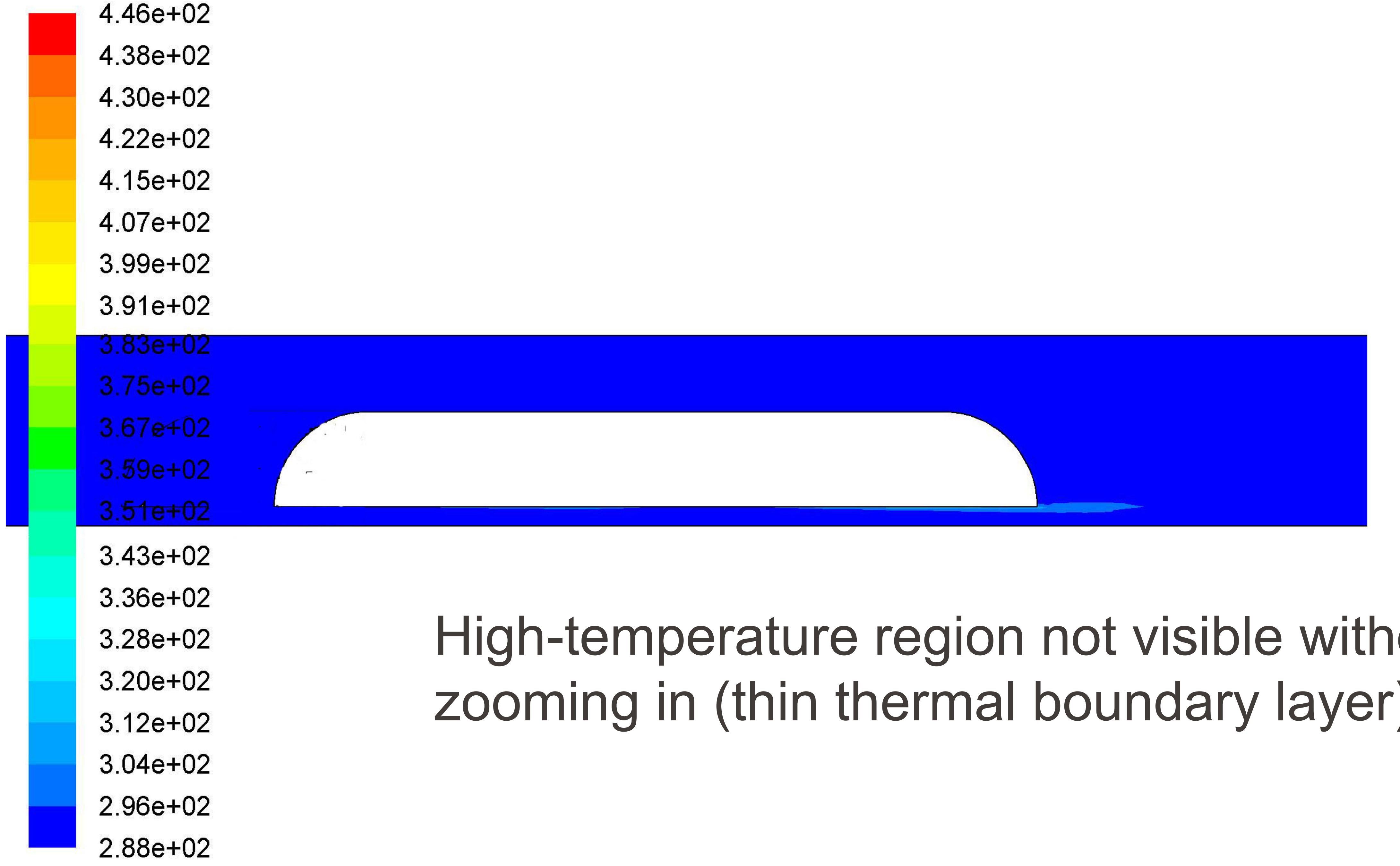
- Contours of vorticity



Wake, strong gradients
(+ oscillations = hint of unsteadiness)

Post-processing

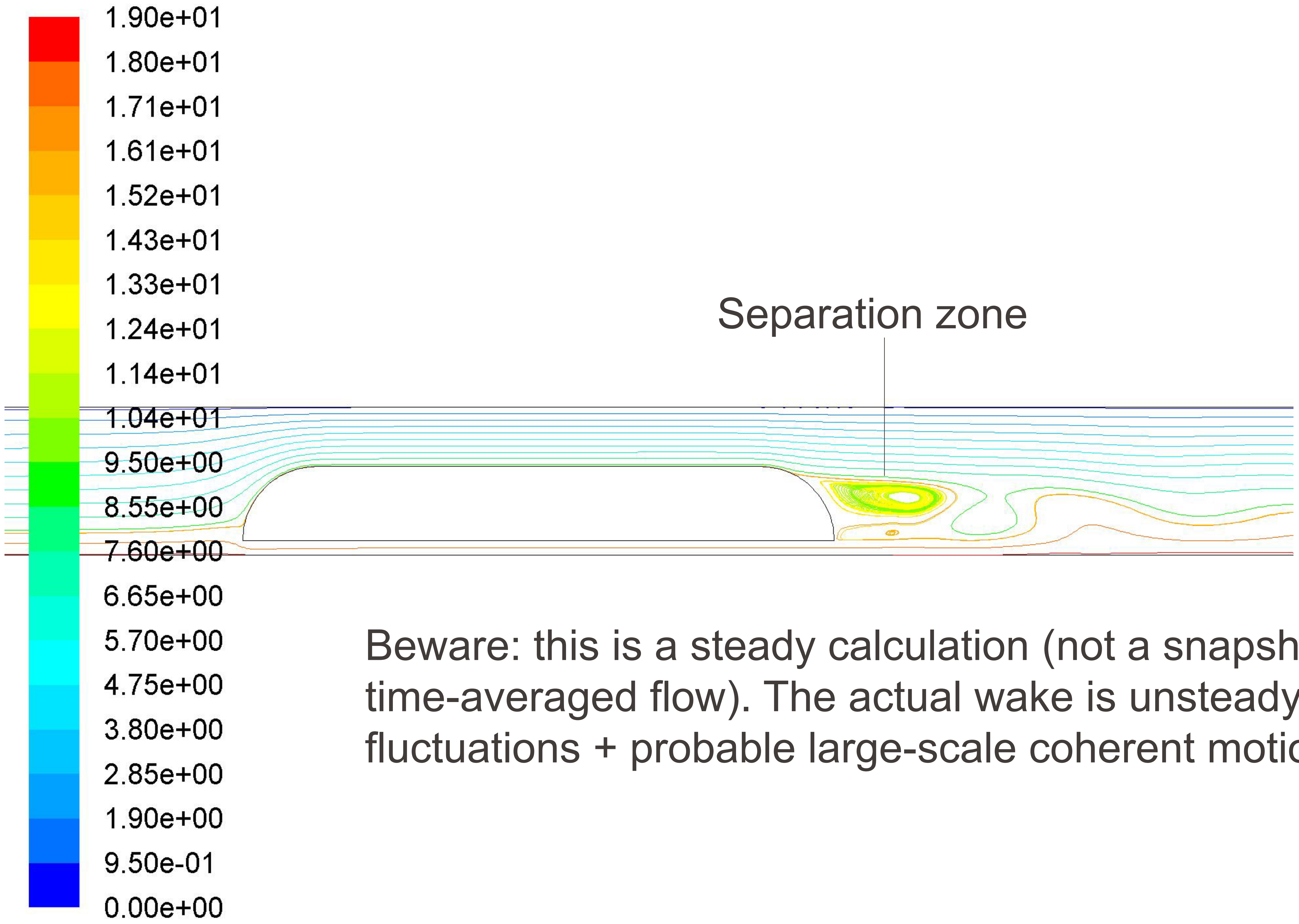
- Contours of temperature



High-temperature region not visible without zooming in (thin thermal boundary layer)

Post-processing

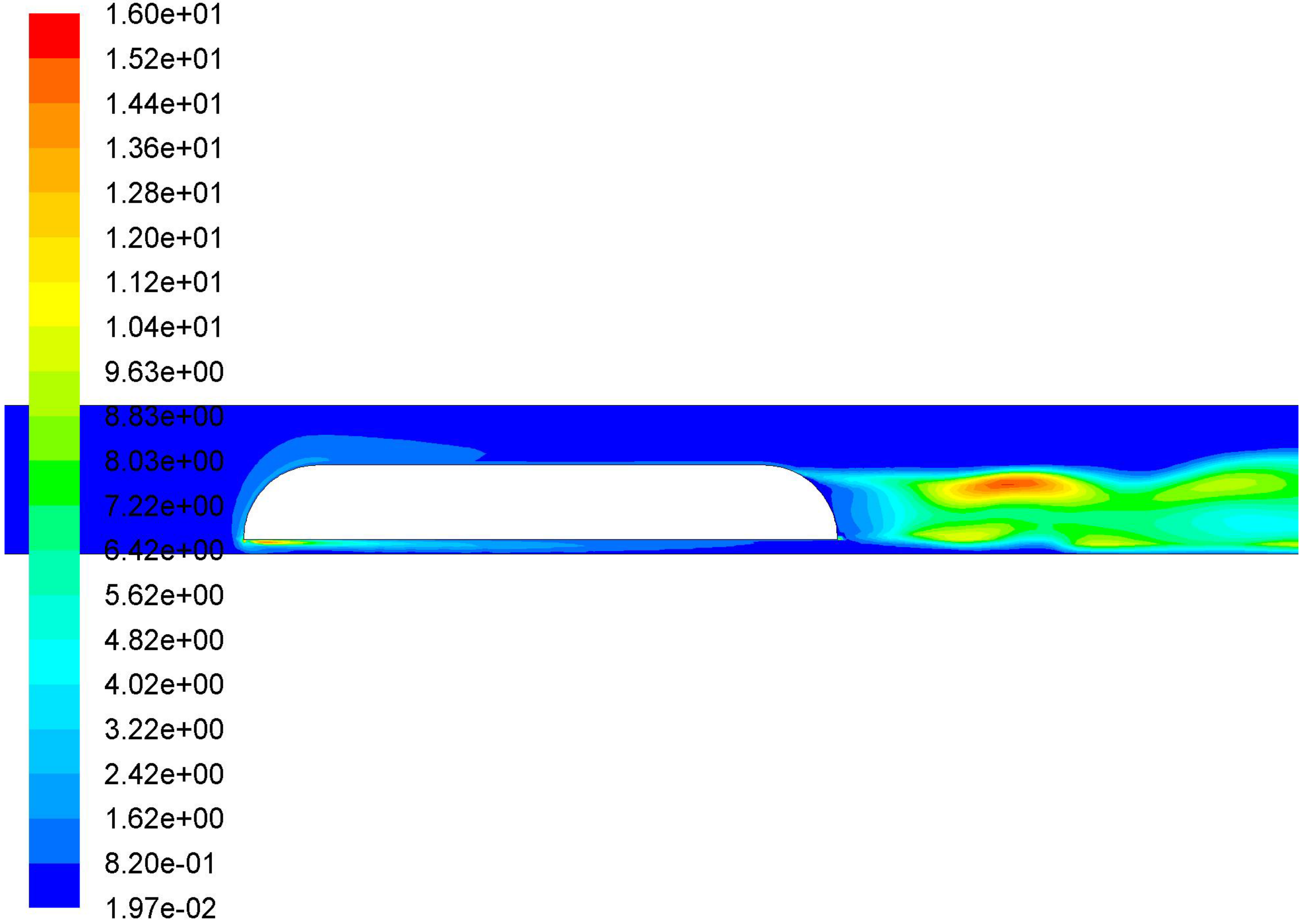
- Streamlines



Beware: this is a steady calculation (not a snapshot, not a time-averaged flow). The actual wake is unsteady (turbulent fluctuations + probable large-scale coherent motion)!

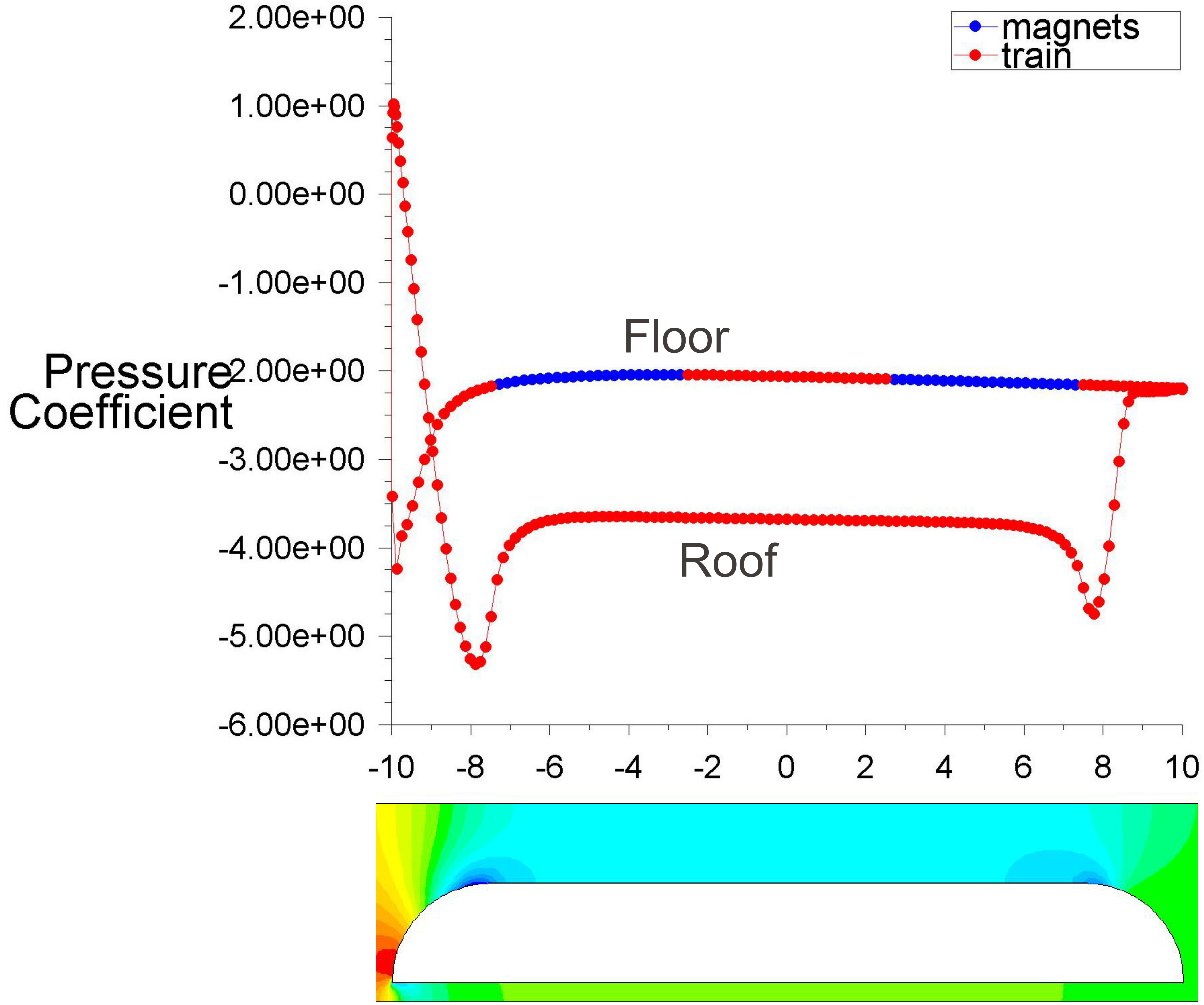
Post-processing

- Contours of turbulent kinetic energy *k*



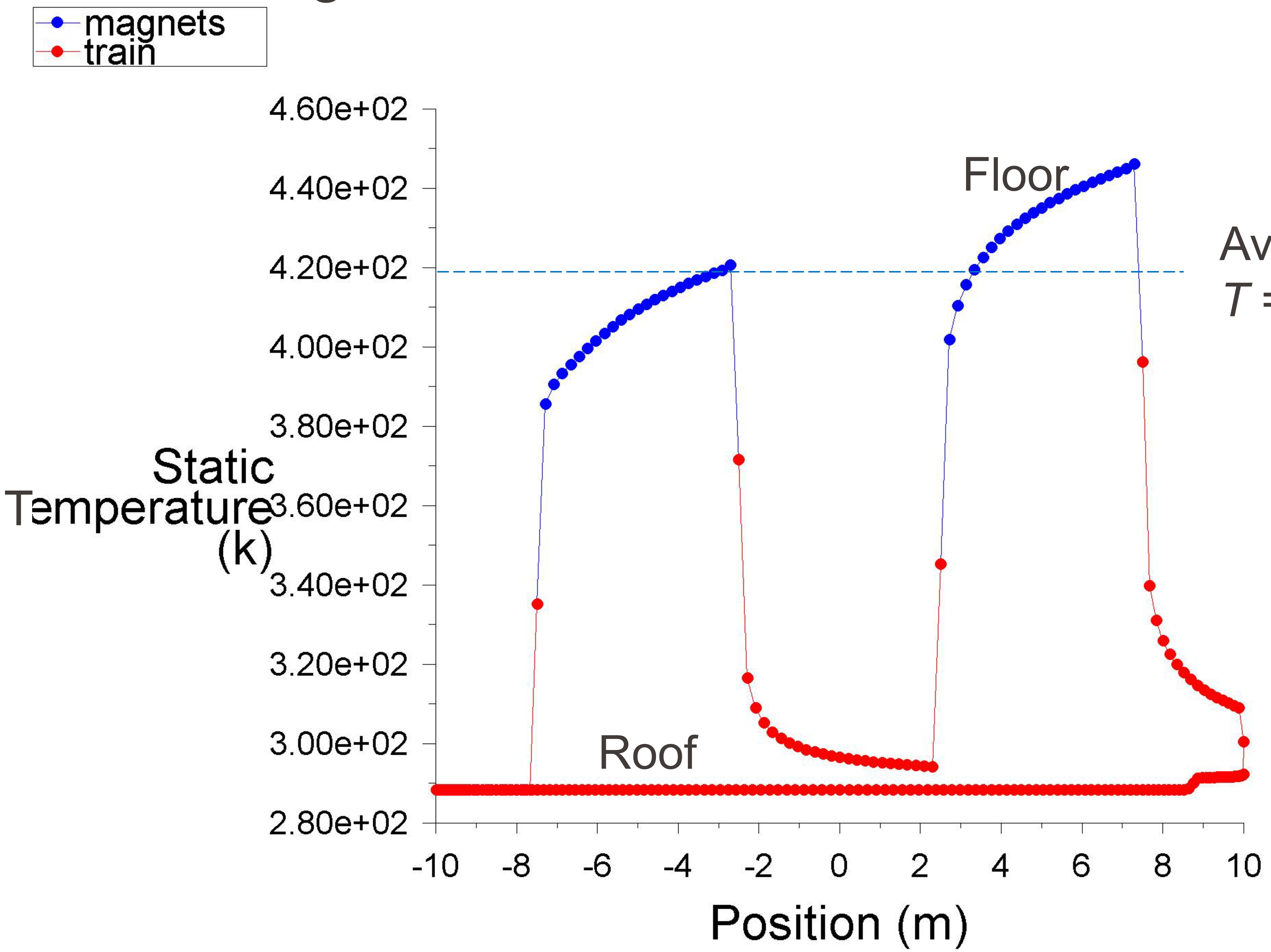
Post-processing

- Pressure coefficient on Maglev wall



Post-processing

- Temperature on Maglev wall



Average over propulsion system:
T = 419 K = 146°C

$T_{in} = T_{\infty}$ = 288 K = 15°C

Numerical Flow Simulation

Global observations

- Physical analysis of the numerical solution:
  - The flow accelerates more over the Maglev than below, producing a region of lower pressure (→ positive lift).
  - Strong wake behind the Maglev, with large velocity gradients; probably unsteady.
  - Flow attached along most of the Maglev, separation in the tail region, associated recirculation zone.
  - Turbulence generation highest in boundary layer / wake due to strong gradients.
  - Thin boundary layer, justifying the use of wall functions in the turbulence model.
  - Wall y^+ around 30, consistent with the choice of BL mesh + wall function. Due to separation, a non-equilibrium wall function has been used.
  - Air temperature raises significantly in the (immediate) vicinity of the propulsion system.

Possible improvements

- In the numerical simulation:
  - Use a longer domain (e.g. $2L$ upstream, $4L$ downstream)
  - Improve quality of the mesh (nose and tail regions)
  - Use enhanced-wall treatment for turbulence model (+ finer BL mesh)
  - Investigate unsteady flow behavior (unsteady simulation)
  - Perform 3D simulation (more realistic geometry, weaker blockage effect)

- In the Maglev design:
  - Increase tunnel diameter compared to Maglev diameter
  - Use different shapes for nose and tail (e.g. elongated nose, etc.)

Your work (optional)

- Perform the flow simulation using ANSYS Workbench and Fluent
  - Use a finer mesh (is the solution converged?)
  - Use a higher-quality mesh (is the convergence rate improved?)
  - Use a longer computational domain (influence of inflow/outflow?)

- Perform a detailed post-processing analysis of the solution.

- Examine the sensitivity of the solution on:
  - the mesh,
  - the turbulence model,
  - the numerical method.