A benchmark study for CFD solvers: simulation of air flow in livestock husbandry

Alfonso Caiazzo (WIAS)
D. Janke (ATB), D. Willink (ATB),
N. Ahmed (ex-WIAS), O. Knoth (TROPOS)

Leibniz Association

ATB Leibniz-Institut für Agrartechnik und Bioökonomie

TROPOS Leibniz Institute for Tropospheric Research

MMS Days 2018@Leipzig

Collaborators

- David Janke, Dillya Willink
- Leibniz Institute for Agricultural Engineering and Bioeconomy: research at the interface of biological and technical systems

Leibniz-Institut für Agrartechnik und Bioökonomie

- Department *Engineering for Livestock Management*: combination of basic and applied research in animal husbandry to improve animal welfare and animal protection

Leibniz Institute for Tropospheric Research

- Oswald Knoth
- Research on atmospheric aerosols, involving experimental investigations and model simulations on different atmospheric scales.

- Alfonso Caiazzo, Naveed Ahmed (ex-WIAS)
- Research group Numerical Mathematics and Scientific Computing, focus on modeling and simulation of fluid, esp. using finite element method

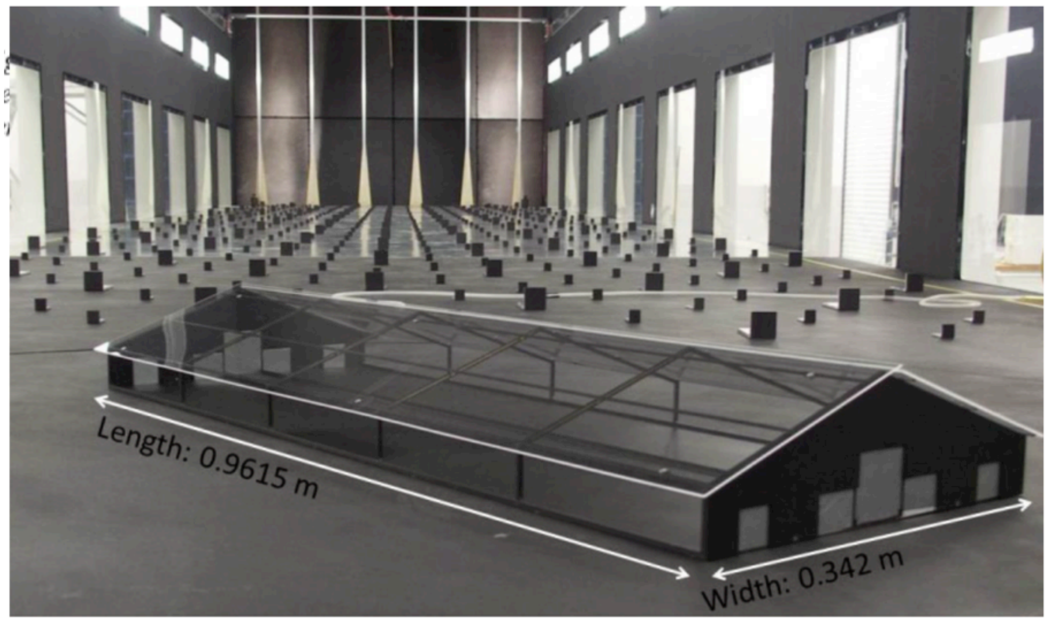
Weierstrass Institute for Applied Analysis and Stochastics

Overview

- **What?**
- **Why?**
- **How?**
- **Preliminary results (Jun – Nov 17)**
- **Conclusions and outlook**

The benchmark problem: (1:100) Windtunnel model

Airflow simulation of livestock husbandry (animal care)



- 1:100 scaled model of an experimental barn in northern Germany
- Windtunnel experimental studies

Motivations & Goals

- Collaboration started during MMS 2017 in Hannover
- Open source mesh generator
- Foster interaction within MMS
- Exploit (interdisciplinary) MMS network as a chance to „learn“ different languages and establish collaboration

ATB

- share knowledge with other institutes
- get better understanding of CFD solvers
- compare open source tools

TROPOS

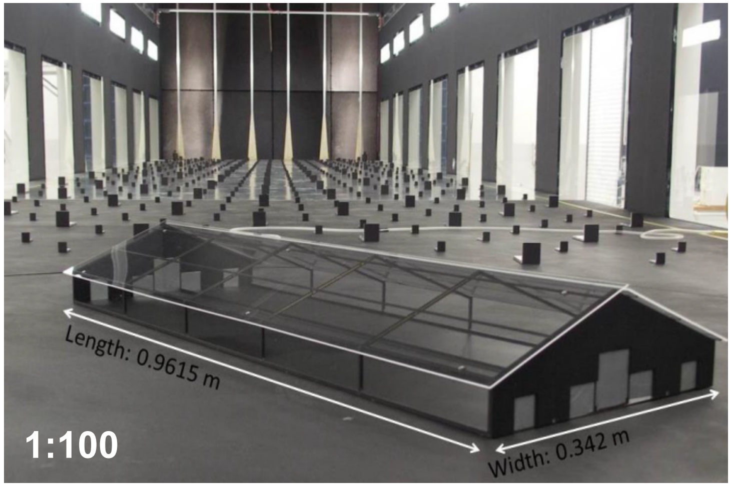
- improve outreach of fluid solver
- test it in different application

WIAS

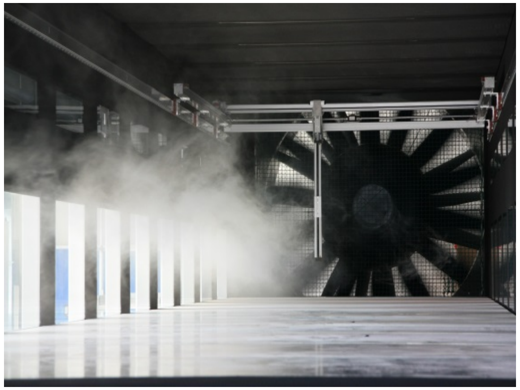
- benchmark the finite element solver against other codes
- compare with against real data
- learn needs of experimentalits

Experimental setup

- 1:100 scaled model of an experimental barn in northern Germany

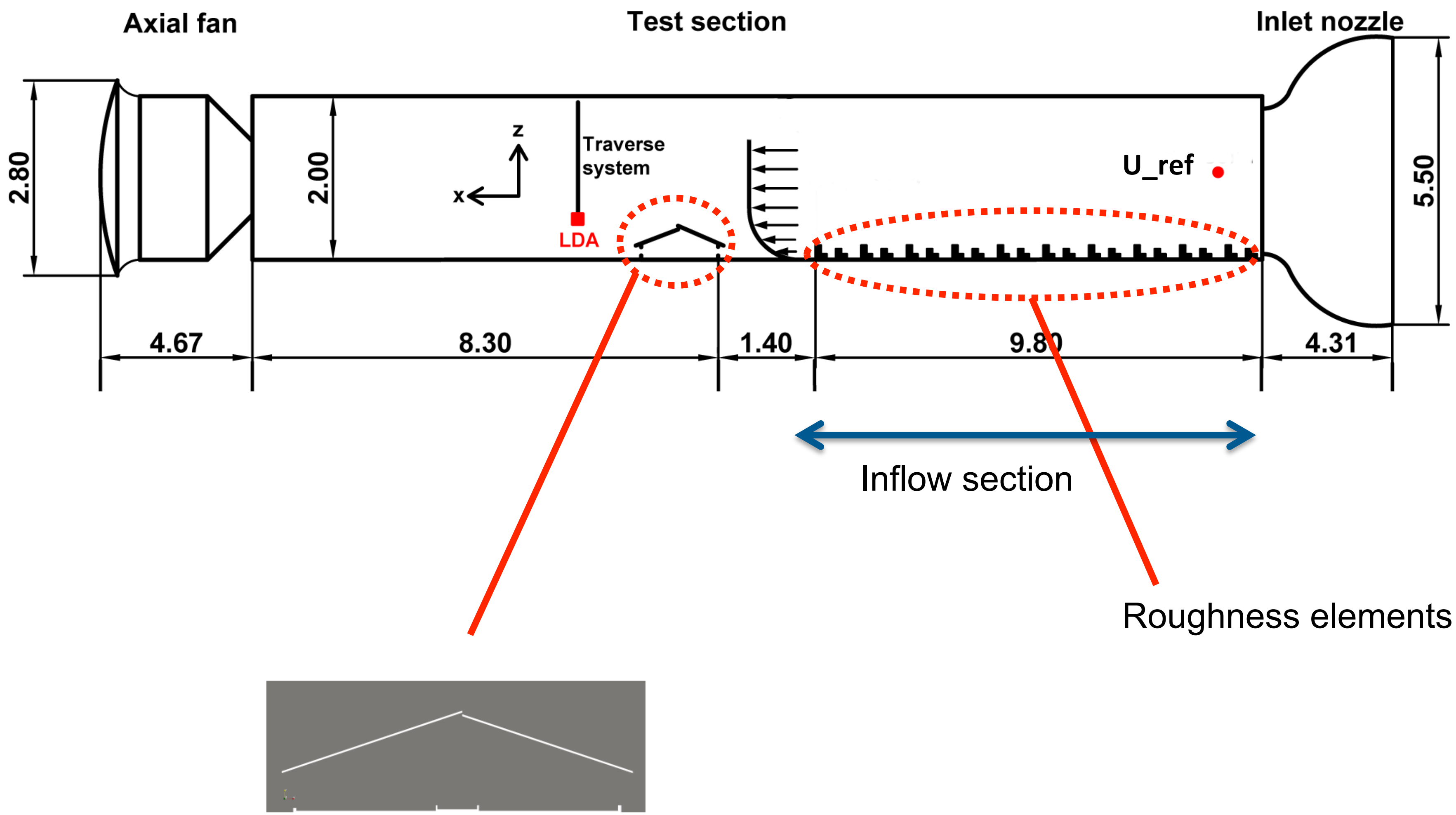


- Fully developed turbulent flow with the use of roughness elements and turbulence generators



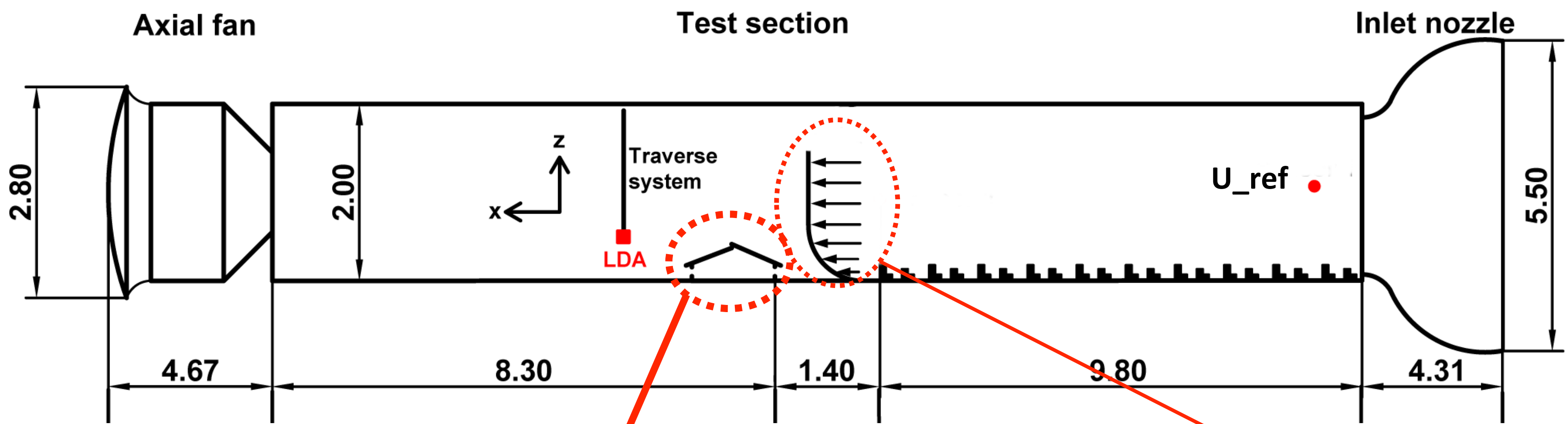
Experimental setup

- ATB large atmospheric boundary layer wind tunnel (ABL-WT)

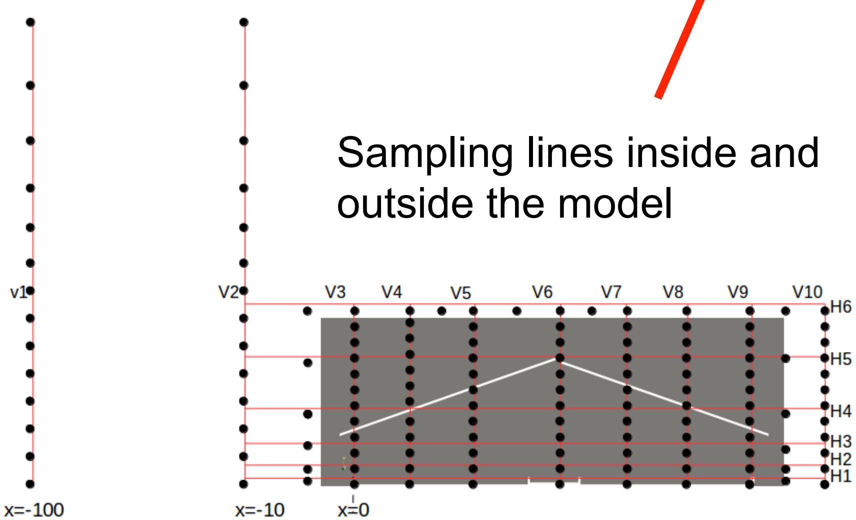


Experimental setup

- ATB large atmospheric boundary layer wind tunnel (ABL-WT)

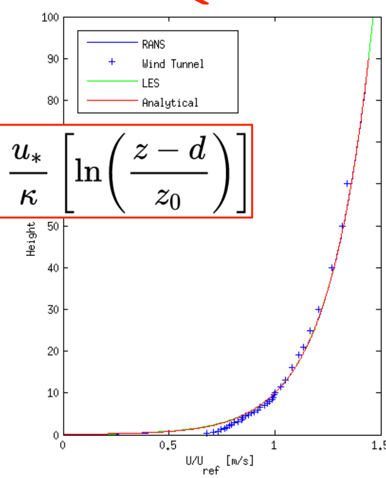


Inflow section



Sampling lines inside and outside the model

Fully developed turbulent flow



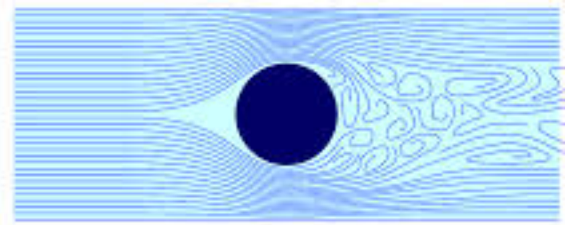
$$u_z = \frac{u_*}{\kappa}\left[\ln\left(\frac{z-d}{z_0}\right)\right]$$

Mathematical Model

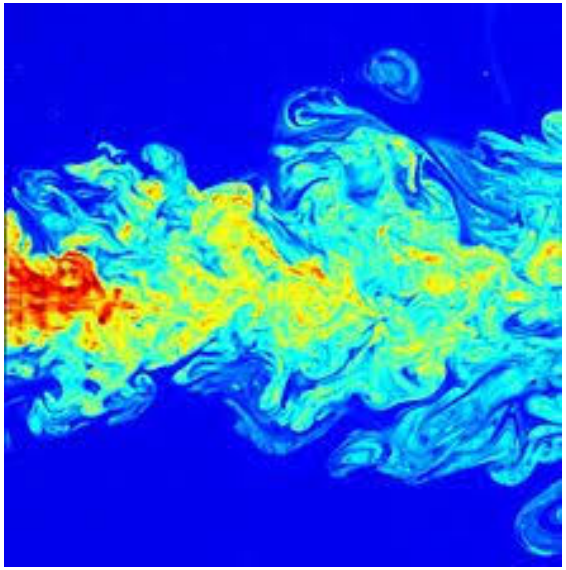
- Navier-Stokes equations (incompressible fluid)

$$\rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot \nu D(\mathbf{u}) + \rho(\mathbf{u} \cdot \nabla)\mathbf{u} - \nabla p = 0$$

$$\nabla \cdot \mathbf{u} = 0$$



- Modeling and simulation of turbulent flows



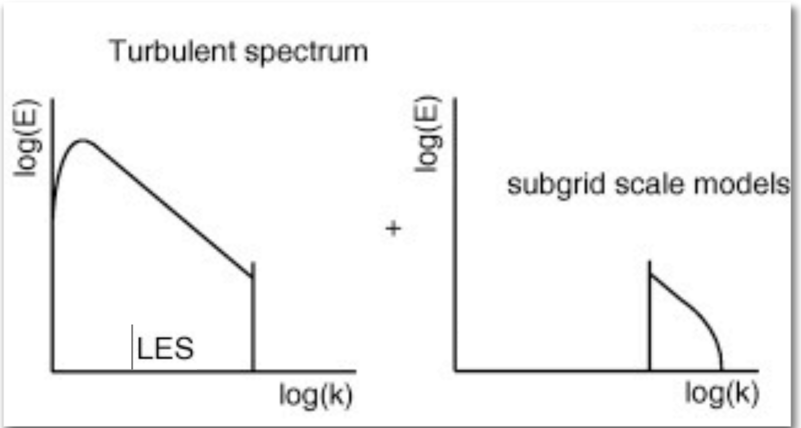
- Direct numerical simulations (DNS): resolve all scales of motion (costly)

- Large-eddy simulations (LES) : focus on large scales, and model the effect of small scales on large scales via modified viscosity

- Variational multiscale method (VMS): variational setting for modeling scale separation and scale interaction

Turbulence modeling: LES (Smagorinsky)

- Large eddies transport most of mass, momentum and energy



- Filter: $\mathbf{u} = \overline{\mathbf{u}} + \mathbf{u}'$

Turbulence model

Smagorinsky:

$$\tau = -2(C_S\Delta)^2\|D(\overline{\mathbf{u}})\|D(\overline{\mathbf{u}})$$

Filter length

Model constant

$$\rho\frac{\partial\overline{\mathbf{u}}}{\partial t} - \nabla\cdot\nu D(\overline{\mathbf{u}}) + \nabla\cdot\tau + \rho(\overline{\mathbf{u}}\cdot\nabla)\mathbf{u} - \nabla p = 0$$

$$\nabla\cdot\overline{\mathbf{u}} = 0$$

Turbulence modeling: Variational Multiscale (VMS)

- Three scales: large, small-resolved, small-unresolved
- Scale-separation and sub-grid model directly embedded into the variational formulation
- Finite element method: natural discrete setting

$$\rho\left(\frac{\partial \mathbf{u}}{\partial t}, \mathbf{v}\right) + (\nu D(\mathbf{u}), D(\mathbf{v})) + \rho((\mathbf{u}\cdot\nabla)\mathbf{u}, \mathbf{v}) - (p, \nabla\cdot\mathbf{v}) + \left(\nu_T\left(D(\mathbf{u}) - G^H\right), D(\mathbf{v})\right) = (\mathbf{f}, \mathbf{v}), \forall \mathbf{v} \in \mathbf{V}$$

$$(q, \nabla\cdot\mathbf{u}) = 0, \forall q \in Q$$

$$\left(D(\mathbf{u}) - G^H, \lambda^H\right) = 0, \forall \lambda^H \in L^H$$

(standard) finite element spaces

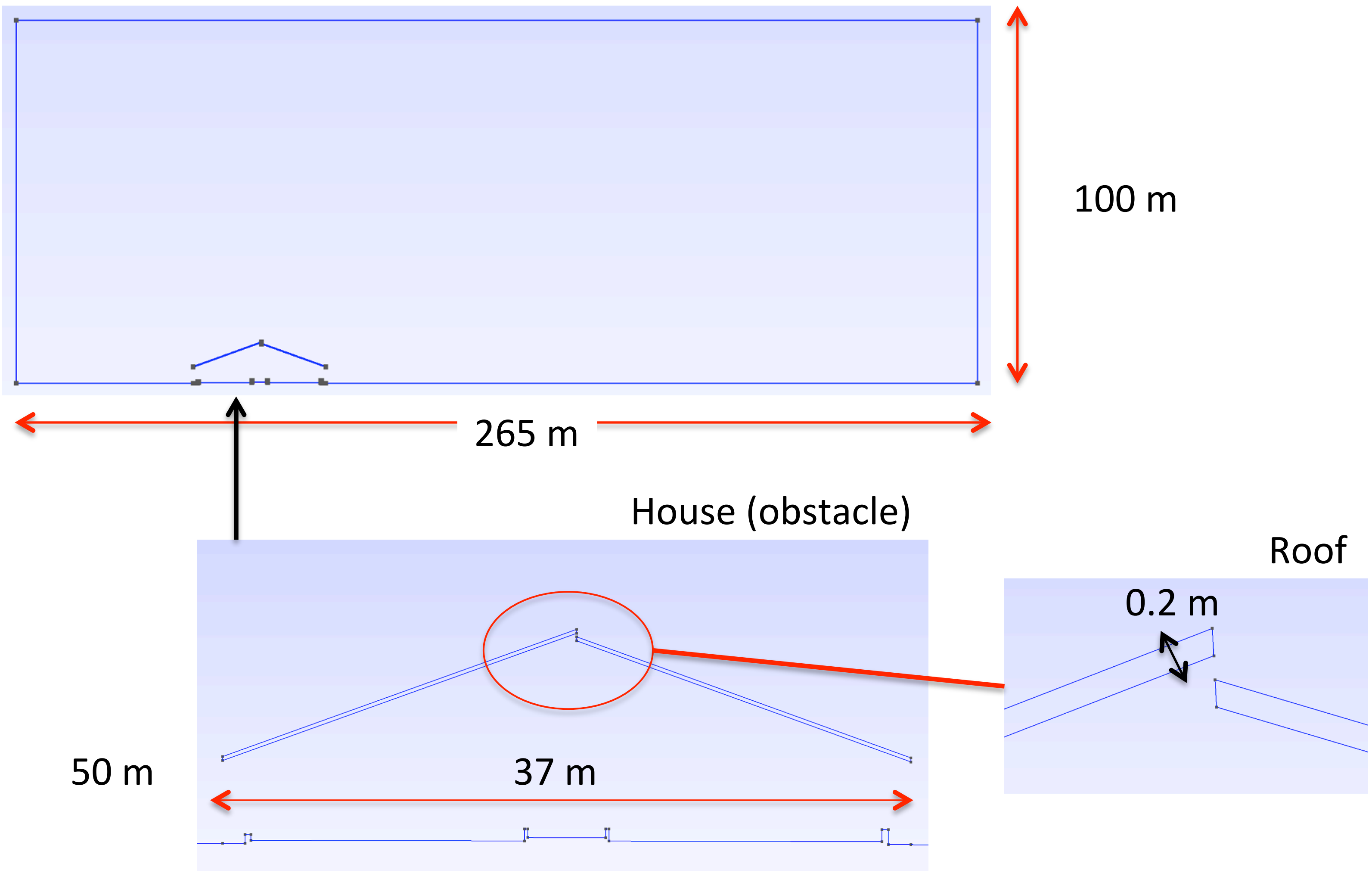
Small scales

Turbulent viscosity (Smagorinsky)

Tensor-valued space of resolved small scales

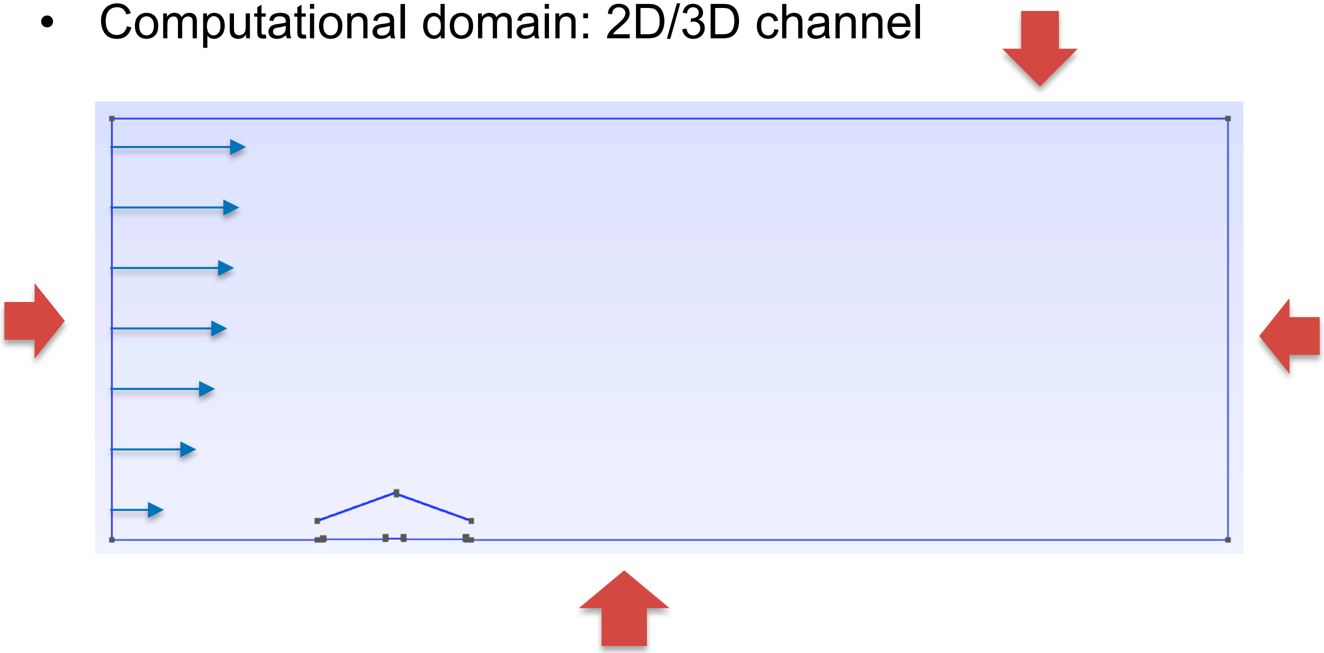
Simulation Setup

- Computational domain:



Simulation Setup

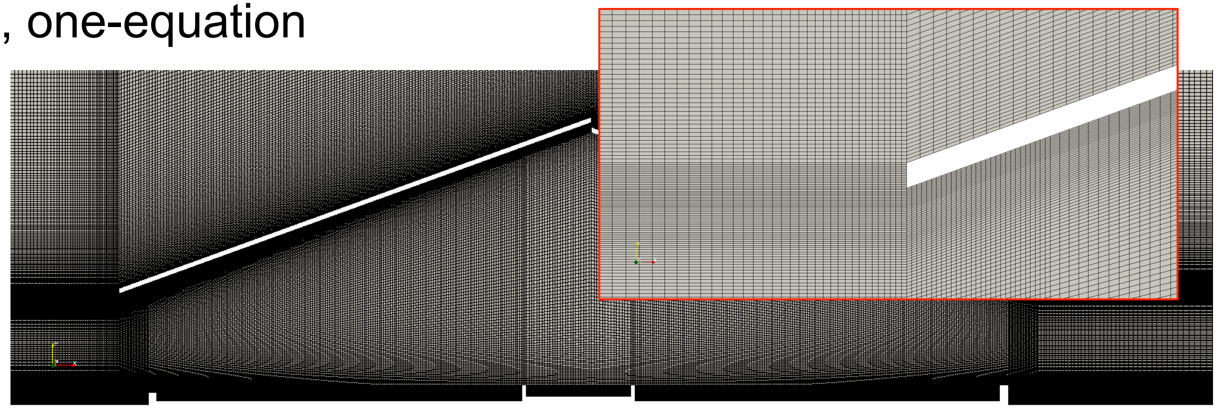
- Computational domain: 2D/3D channel



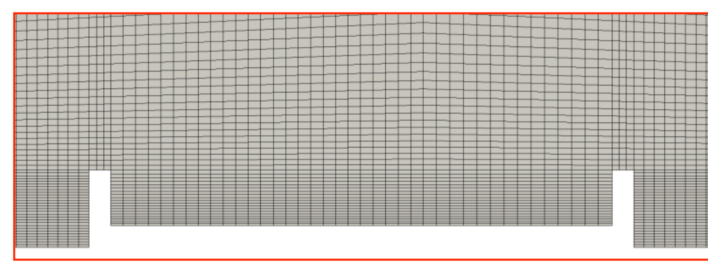
- Boundary conditions:
    - Prescribed inlet profile (windtunel measurements)
    - Do-nothing condition on open boundary
    - No-slip/friction model on the bottom
    - Slip condition on the top
- Time interval: 0 to 1500 seconds (then compute temporal average)

Solver #1: OpenFoam

- Open source CFD (developed by Open CFD)
- Finite Volume, C++
- Widely used across most areas of engineering and science, suitable for different flow regimes (esp. Incompressible and turbulent)
- Turbulence model: LES, one-equation eddy viscosity model



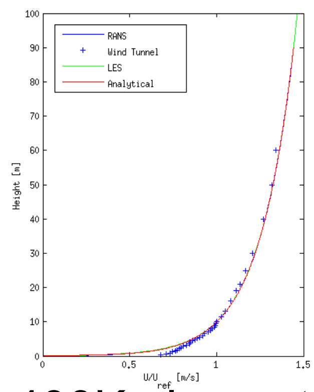
- Block-structured 3d hexahedral meshes
- Computational mesh: 280K cells, about 500K nodes
- Time discretization: explicit backward method, second order, adaptive time step (CFL<0.8)

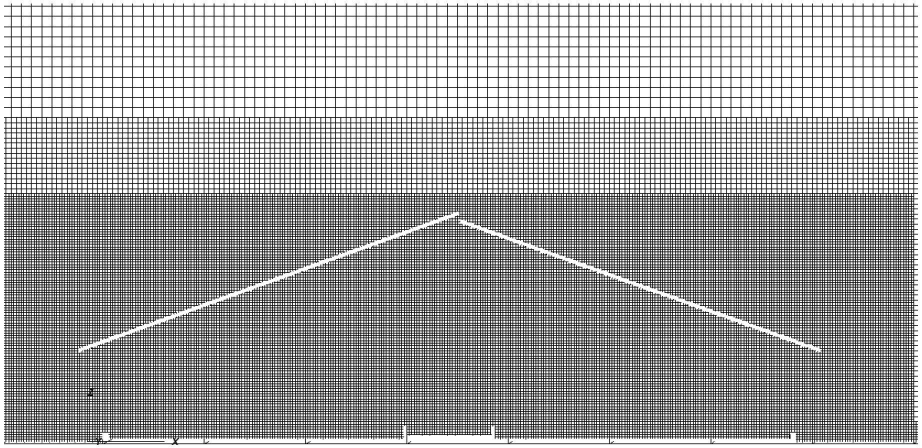


- One Simulation: up to 4 days on 32 CPU

Solver #2: ASAM

- All Scale Atmospheric Model, developed by O. Knoth (TROPOS)
- Compressible and incompressible Navier-Stokes, suitable for different flow regimes
- FORTRAN, Finite volume on Block-Cartesian meshes
- Cut-cell approach for internal boundaries
- Time integration: Rosenbrock W method (implicit, time step 0.01s)
- Turbulence model: LES, Smagorinsky
- No-slip boundary condition: use of a *wall-function* to account for boundary layer

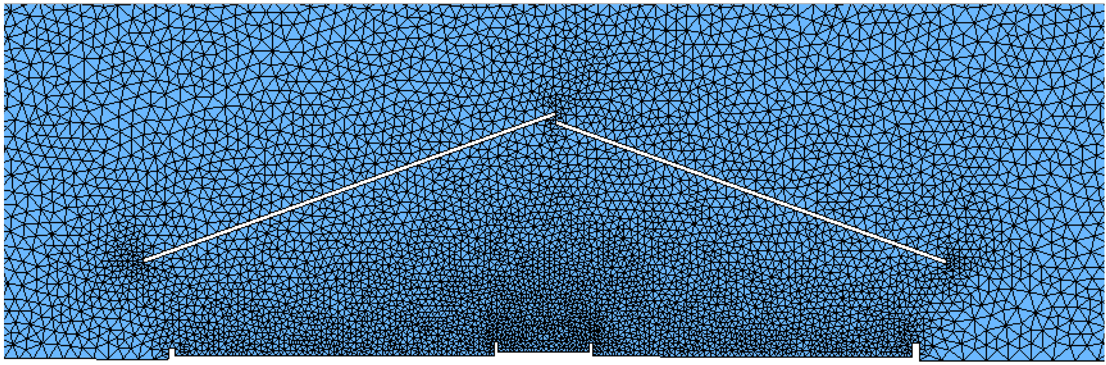




- Computational mesh: 400K elements

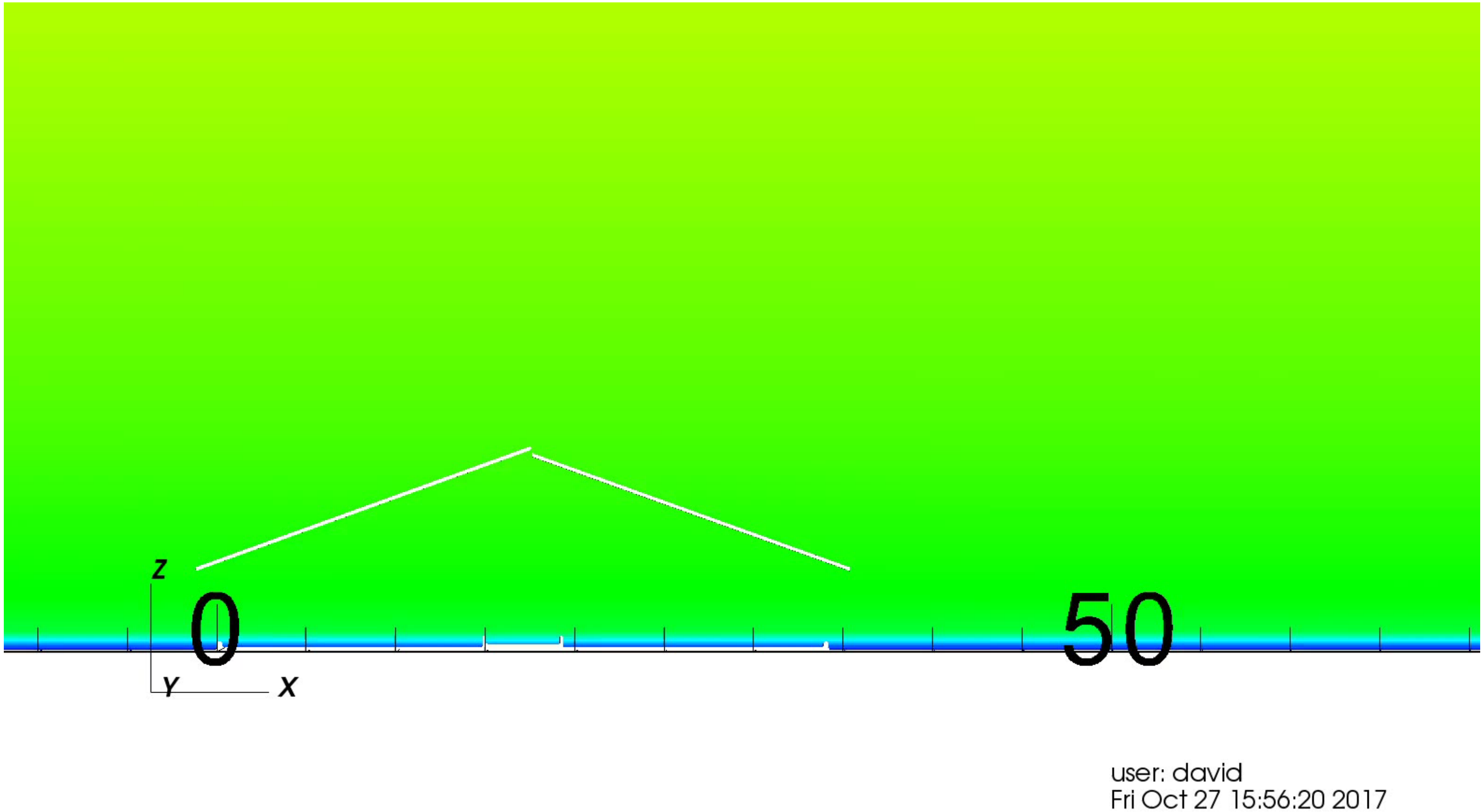
Solver #3: ParMooN

- Parallel mathematics and object-oriented numerics (WIAS, V. John group)
- Focus on flow and trasport (convection-dominated) problems, stabilized finite elements, turbulence modeling
- Finite Element Solver, C++
- Several available options for: finite element spaces (2D and 3D), time discretization, non-linear iteration, direct and iterative linear solvers
- Turbulence Model: Variational Multiscale
- P1/P1 stabilized finite elements
- Computational mesh: 80K triangles, 40K nodes
- Time discretization: 2nd order BDF, time step 0.01 s
- Backflow stabilization on open boundary



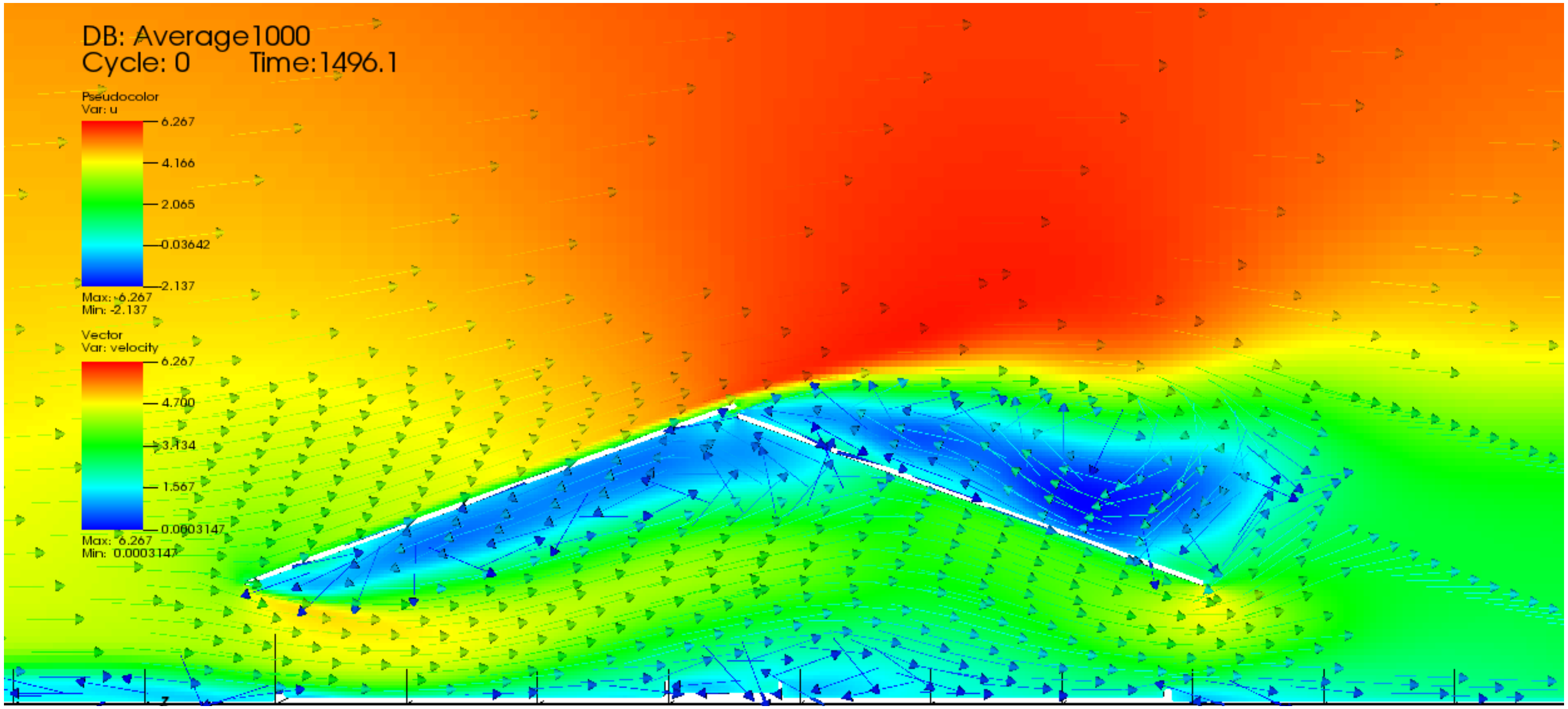
Numerical Results - ASAM

- Velocity magnitude



Numerical Results - ASAM

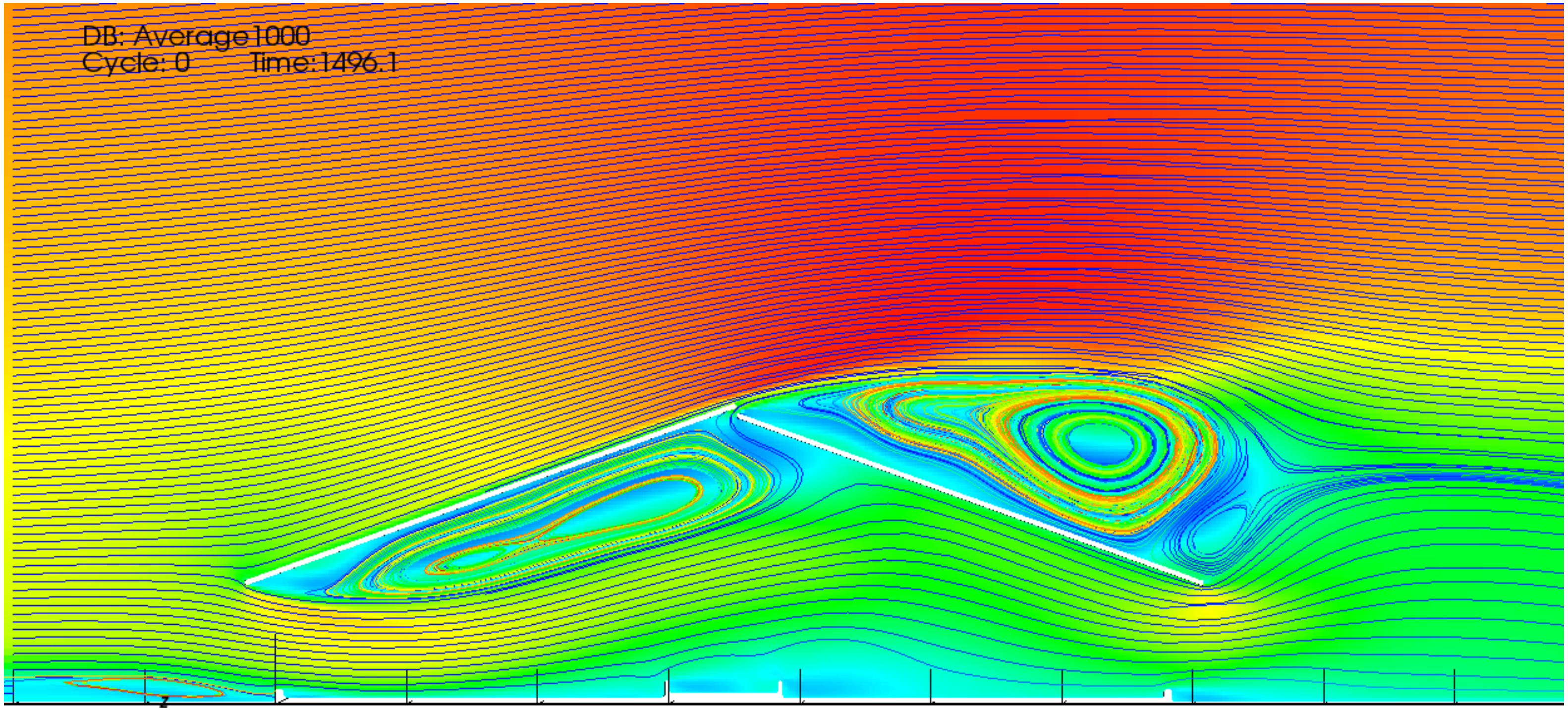
- Velocity vector (mean)



TROPOS Leibniz Institute for Tropospheric Research
ATB Leibniz-Institut für Agrartechnik und Bioökonomie
WIAS
Leibniz Association

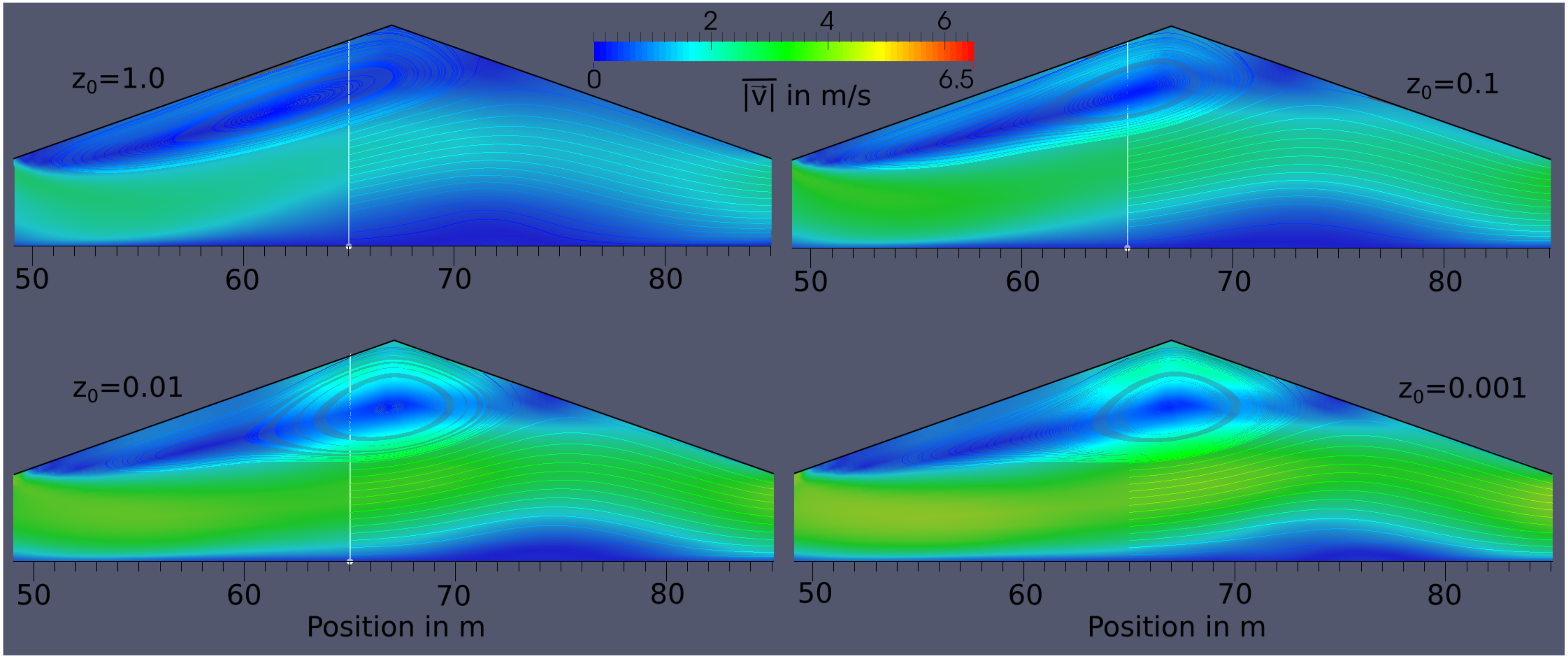
Numerical Results - ASAM

- Streamlines



Numerical Results - ASAM

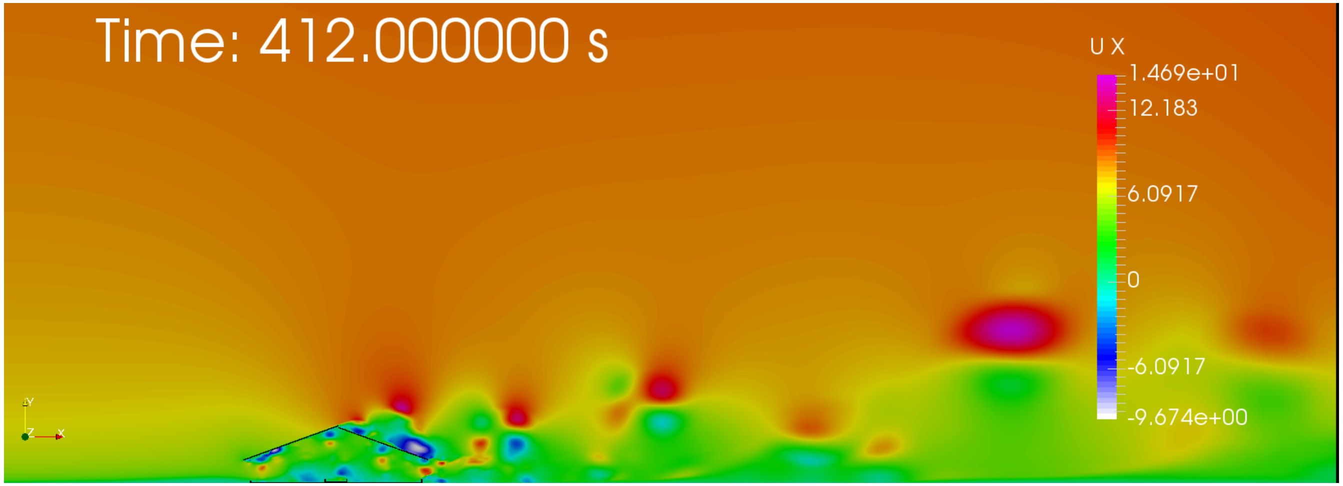
- Effect of wall-function parameter



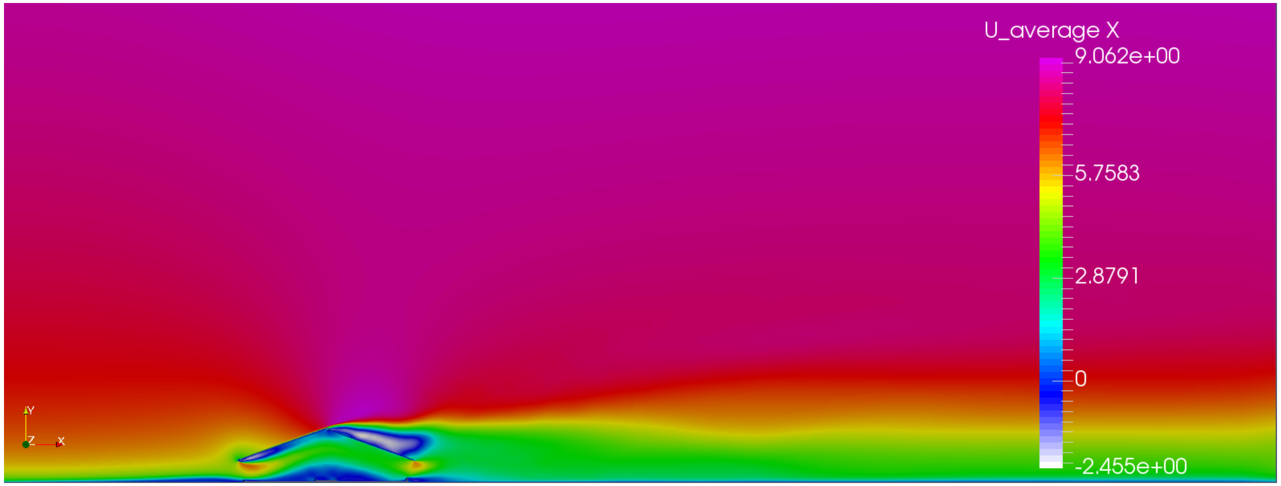
Numerical Results - OpenFOAM

Numerical Results - OpenFOAM

- Snapshot of flow velocity (x-component)

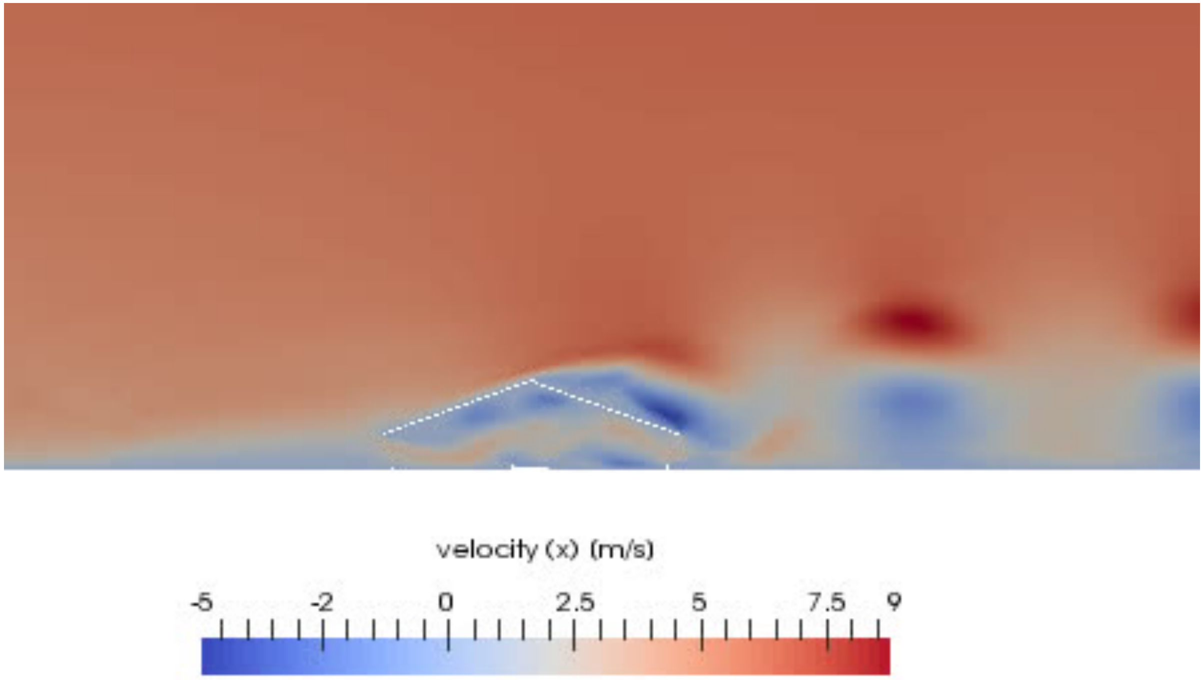


- Average velocity (x-component)

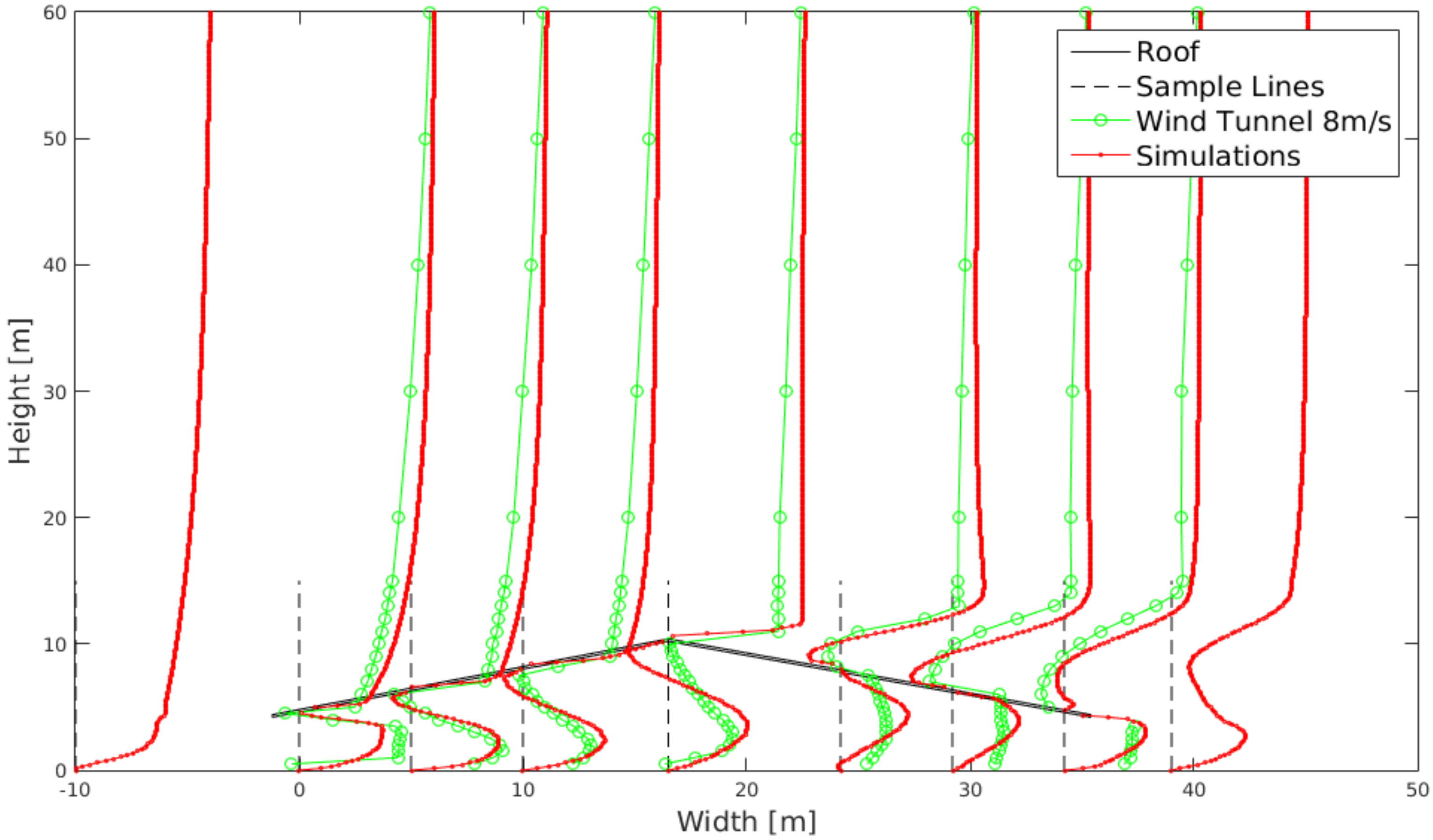


Numerical Results - ParMooN

- Velocity magnitude (zoom near the house)

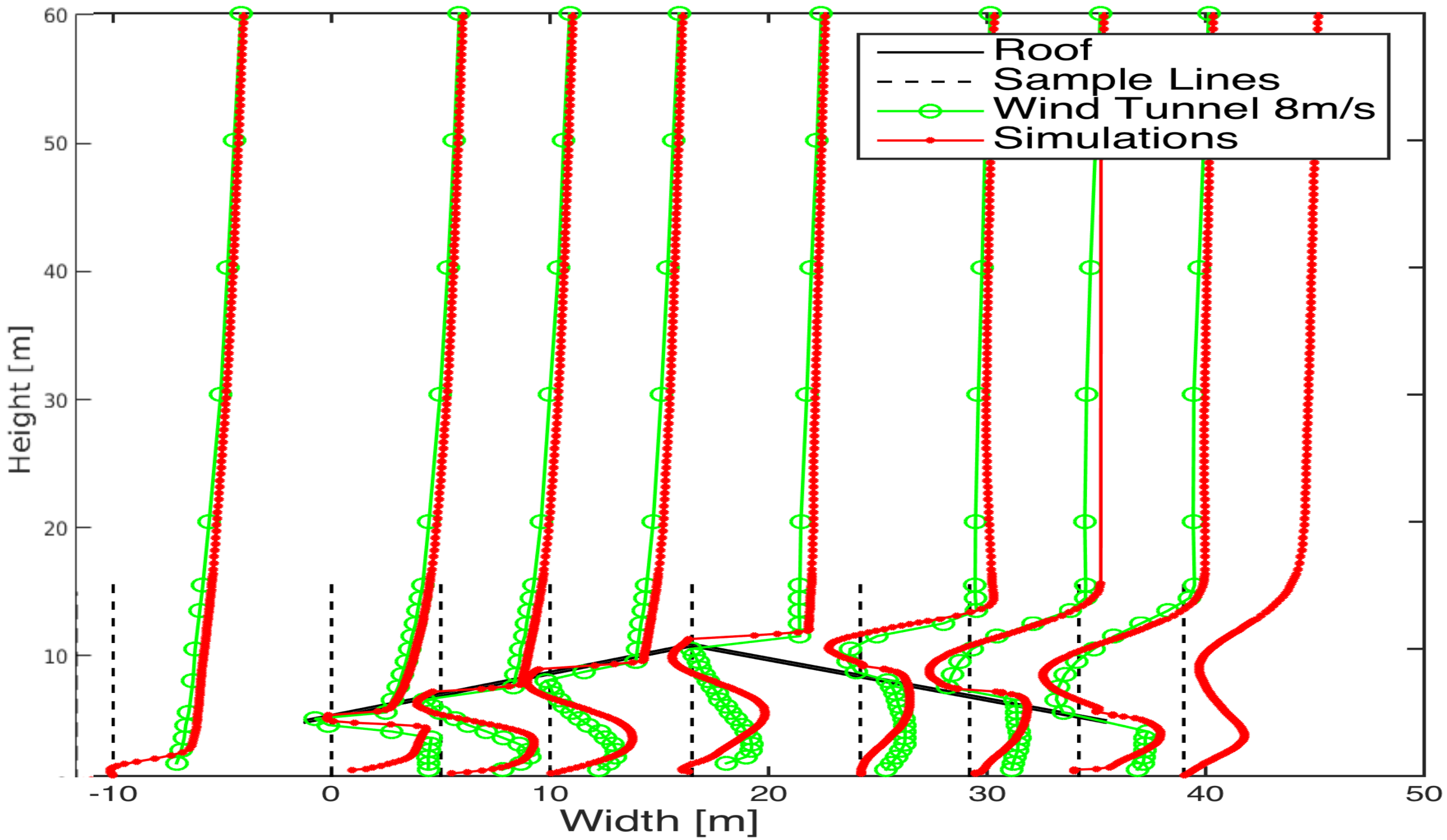


Numerical Results (sample lines - OpenFOAM)

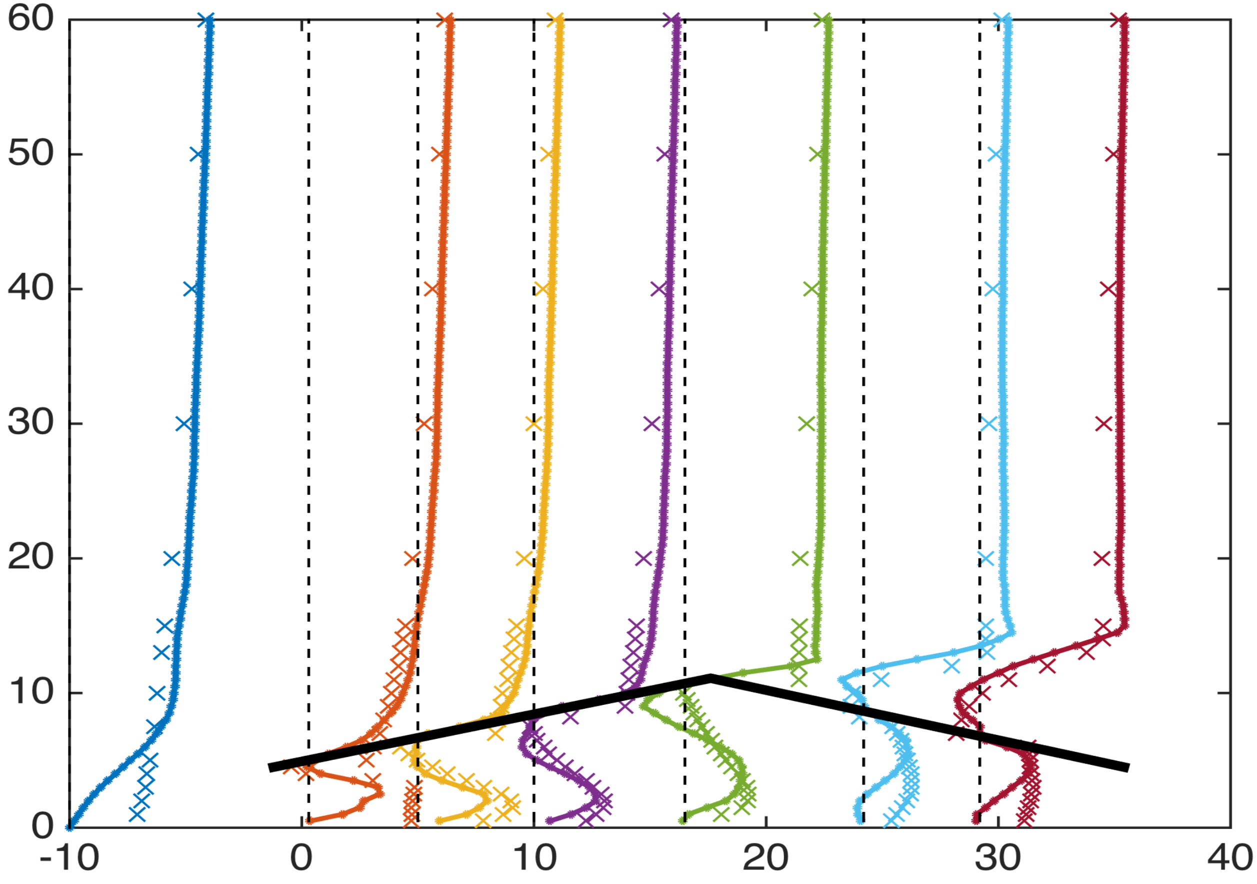


Leibniz Institute for Tropospheric Research
Leibniz-Institut für Agrartechnik und Bioökonomie
Leibniz Association

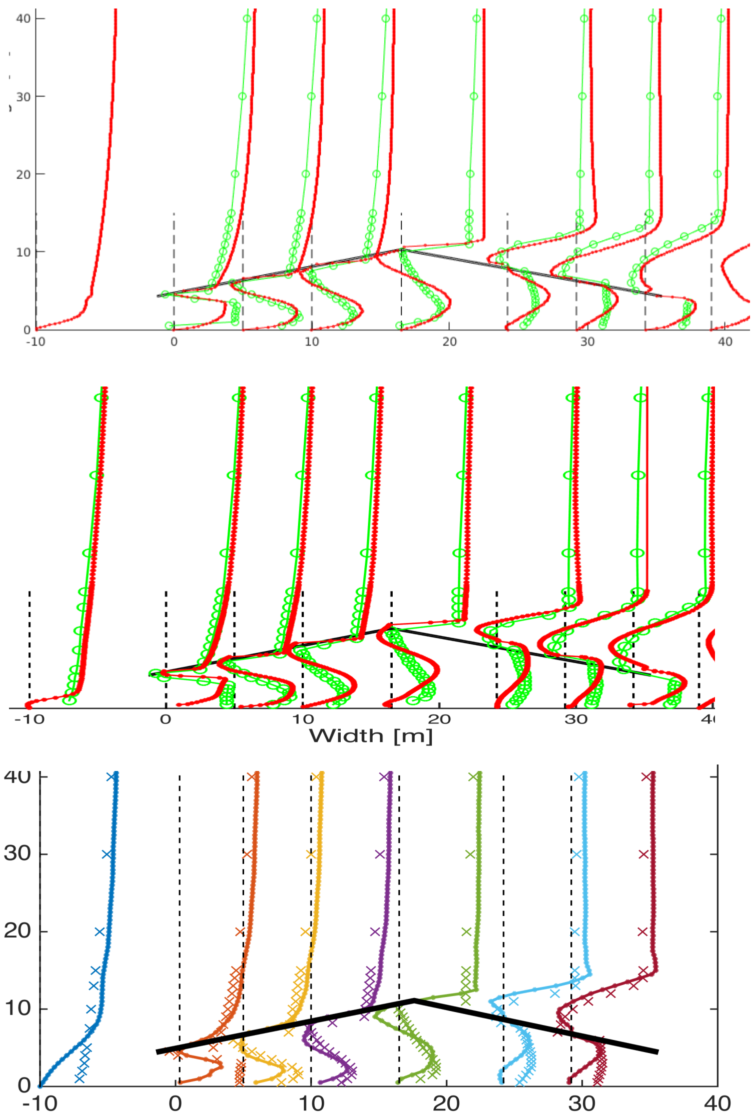
Numerical Results (sample lines - ASAM)



Numerical Results (sample lines - ParMooN)



Numerical Results (remarks)



- Good overall agreement
- Missing: boundary layer
- **ASAM**: more flexible physical modeling, better approximation of boundary layer
- **ParMooN**: less CPU time (due to better time discretization and unstructured mesh)

Open issues/Outlook

1. Simulate inflow (boundary layer) w/o obstacle

   - Reproduce the flow behavior in the inflow section (development of turbulence, boundary layer)
   - Test friction velocity models (wall functions), tune model parameters

2. Simulate different scales of the problem

   - Numerical simulation at the windtunnel scale (1:100)
   - Better understanding of non-linear effects

3. Joint publication (concerning the benchmarking of open source software for the considered application)

Conclusions

- **Benchmark problems are important:**
  - A good benchmarking of existing methods is as relevant developing new methods
  - Key for reproducible research

- **Benchmark might be more complex than expected:** we have to learn to talk to each other

- **Benchmark results are always good:** If the experimental data are not fully matched, the study provides hint about how to improve (mathematical, computational, physical) modeling

- **Benchmark studies are always ongoing:** A benchmark study is made to be continuosly updated.

- MMS network provided a necessary framework for this collaboration, and we are happy to share more results with other institutes

THANK YOU!