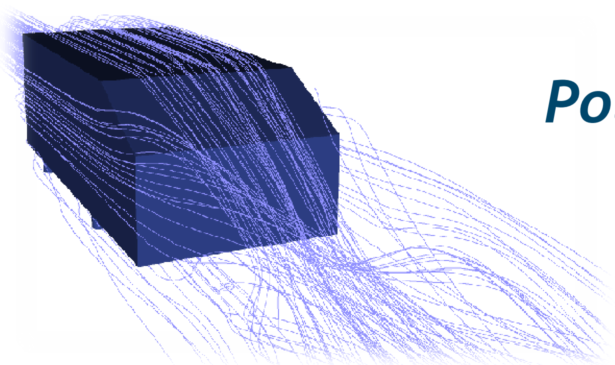
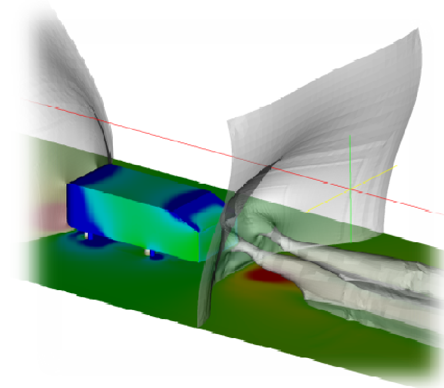


3rd OpenFOAM Workshop
Politecnico di Milano, Milan, ITALY
10-11 July 2008



Investigation of the flow around the Ahmed body using RANS and URANS with various turbulence models

Sebastian MÖLLER[1] , Daniele SUZZI[1], Walter MEILE[2]

[1] The Virtual Vehicle Competence Center (ViF), Graz, Austria

[2] Graz University of Technology, Institute of Fluid Mechanics and Heat Transfer (ISW), Graz, Austria

ISW

K plus Kompetenzzentrenprogramm

Eine Förderinitiative des Bundesministeriums für Verkehr, Innovation und Technologie (BMVIT).
Gefördert mit Mitteln der FFG, des Landes Steiermark und der steirischen Wirtschaftsförderung (SFG)

Ein Kompetenzzentrum der TU Graz

Graz University of Technology

MEMBER OF NAFEMS

autosim PROJECT PARTNER

Overview

Introduction

Test case definition

Numerical method

Results

Conclusions

Overview

The Virtual Vehicle Competence Center

Introduction

Test case definition

Numerical method

Results

Conclusions

Introduction

Objective

- Simulation of flow around the bluff „Ahmed body“ as an example for the application of OpenFOAM in **industrial vehicle aerodynamics**

Method

- **Variation** of meshes, turbulence models, convection schemes and simulation methods (steady/unsteady) to find good overall settings
- Simulation of cases with various angles of attack
- Evaluation of the results against measurement data and results of calculations with other CFD programs
- Evaluation of different solver settings for optimal use of resources

Overview

Introduction

Test case definition

Numerical method

Results

Conclusions

Ahmed body

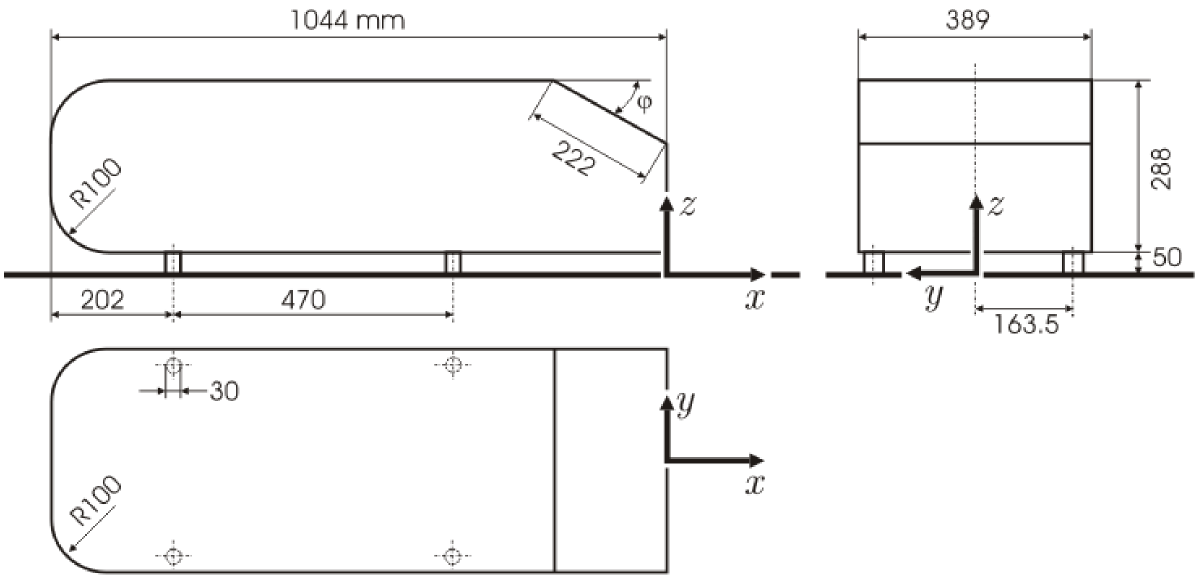
Bluff body for validation of measurements and simulations in vehicle aerodynamics

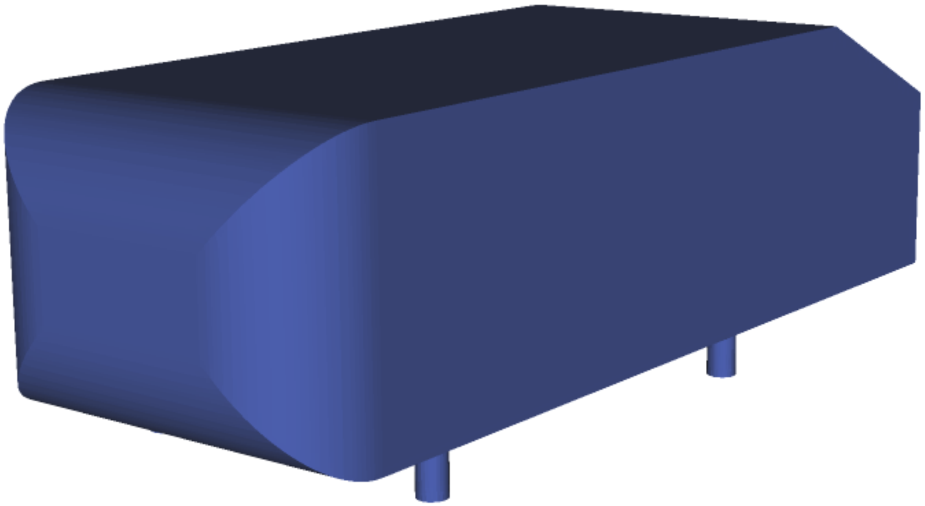
- simple geometry
- typical stream shapes
- experimental data exist, e.g.:

❖ S.R. Ahmed, SAE Paper 840300

❖ S. Becker, H. Lienhart,C. Stoots, ERCOFTAC workshop 9.4 (2000): LDA-Measurement

❖ W. Meile (ISW of Graz UT), 2007, forces and pressure distribution





Overview

Introduction

Test case definition

Numerical method

Results

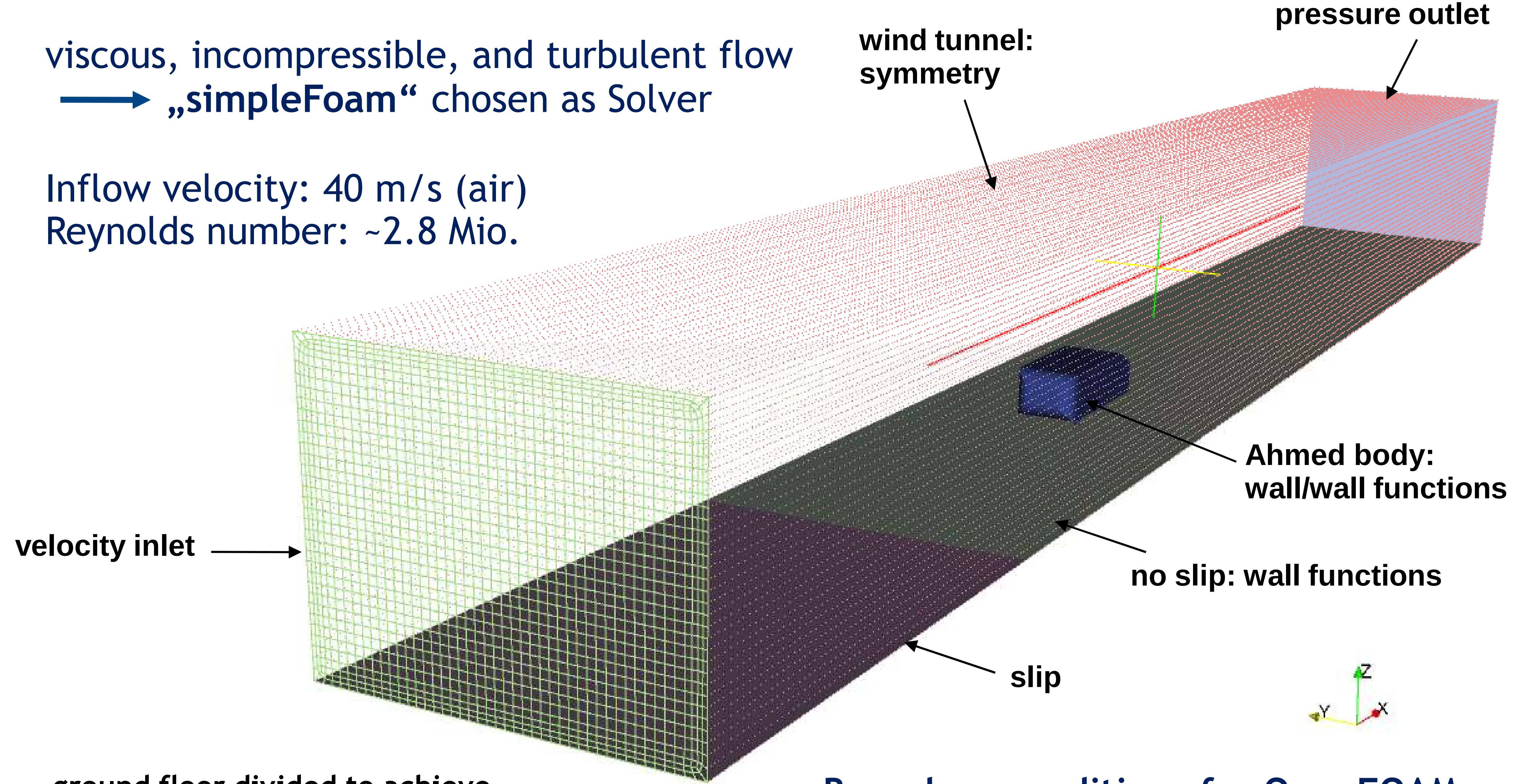
Conclusions

Simulation environment

Geometry of the wind tunnel is given by a recommendation of Ercoftac: 15 x 1.87 x 1.4 m^3

viscous, incompressible, and turbulent flow
→ **„simpleFoam“** chosen as Solver

Inflow velocity: 40 m/s (air)
Reynolds number: ~2.8 Mio.



ground floor divided to achieve correct boundary layer thickness

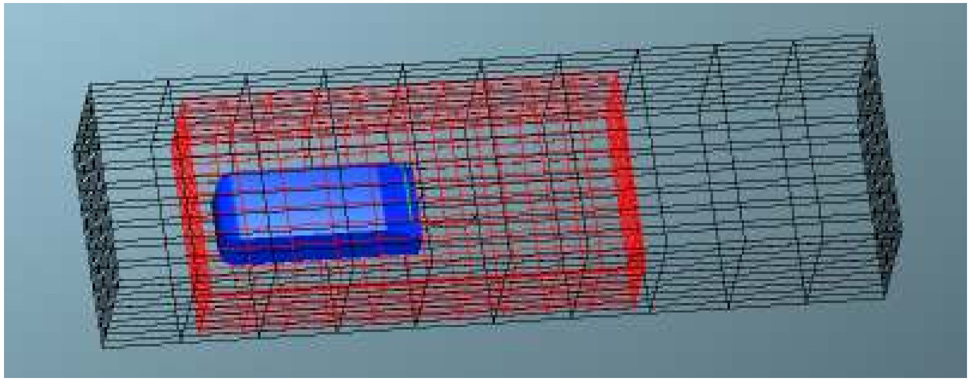
Boundary conditions for OpenFOAM

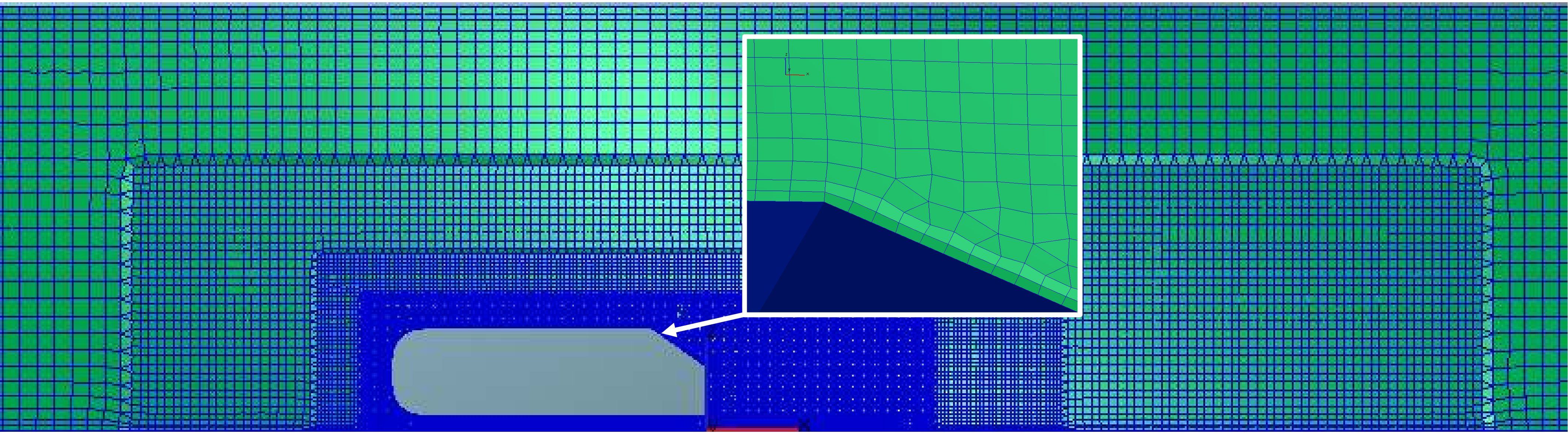
Meshes

Commercial meshing program SPIDER

→ hybrid, hexahedron-dominant meshes with prisms and tetrahedrons in the transition and boundary layers

→ stepwise refinement determined by geometrical boxes around the Ahmed body





Variations

Geometry	25° slant 35° slant	Simulation and comparison of all cases with straight inflow (0° angle of attack) → 12 variations *(only best results are presented)*
Meshes	very coarse (~1 Mio. cells) coarse (~2-2.5 Mio. cells) fine (~4.7 Mio. cells)	
Turbulence model	SST („kOmegaSST“) RKE („realizableKE“)	
Convection scheme	Upwind (1st order) Linear limited SFCD MUSCL	Simulations with different convection schemes only for a reference case → 4 variations
Angle of attack	0° 9° 15°	Simulations with different angles of attack only for a reference case → 3 variations

Personal "best practice" for the numerical settings

	OpenFOAM version 1.4.1
Flow solver	simpleFoam (modified)
Turbulence model	RKE
Schemes for discretization	• „Gauss SFCD" or „Gauss SFCDV" for convection • „Gauss linear" or „corrected" for other operators
Linear solver	• GAMG (Multigrid) for pressure with: smoother = GaussSeidel; nCellsInCoarsestMesh = number CPU's x 40-50 • PbiCG (conjugated gradients) for other variables
Solver tolerance	1.0e-07 for all variables
Relaxation coefficients	• 0.3 for pressure • 0.5 for other variables
Nonorthogonal correction	number of correctors for an iteration 1-3

Overview

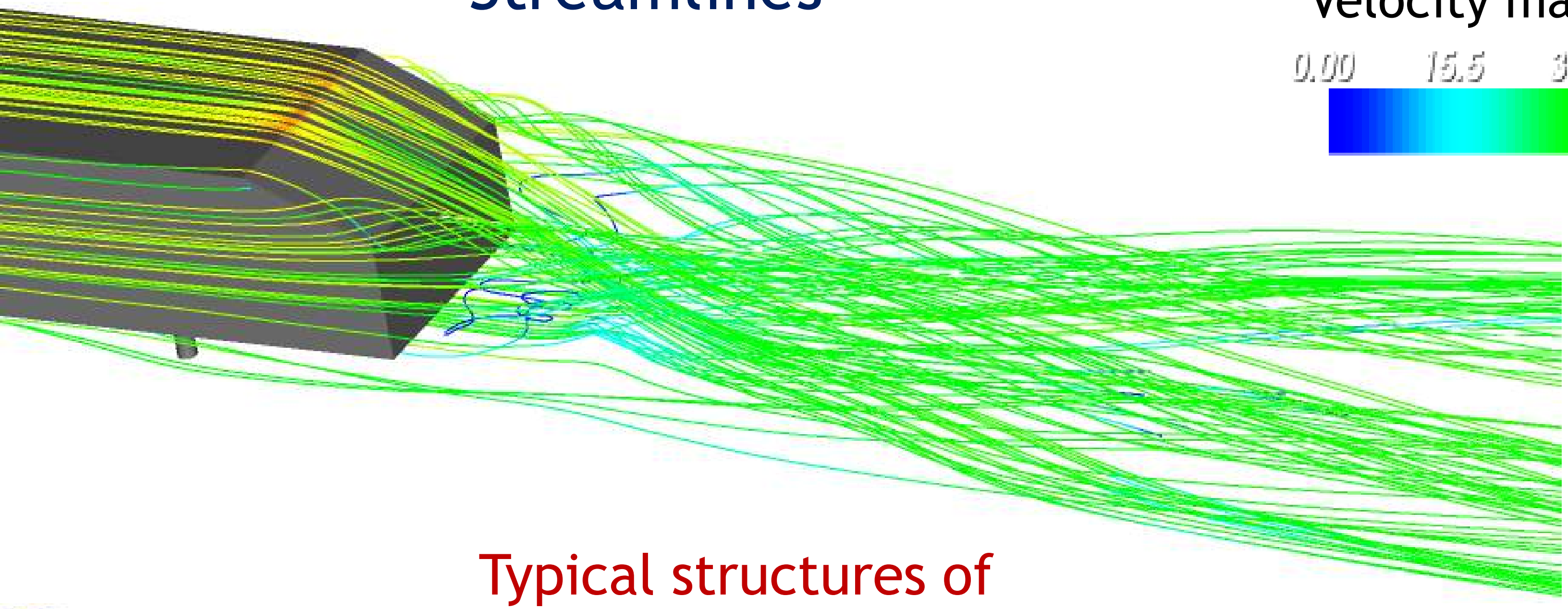
Straight flow (1/2)

The Virtual Vehicle
Competence Center

Streamlines

Velocity magnitude [m/s]

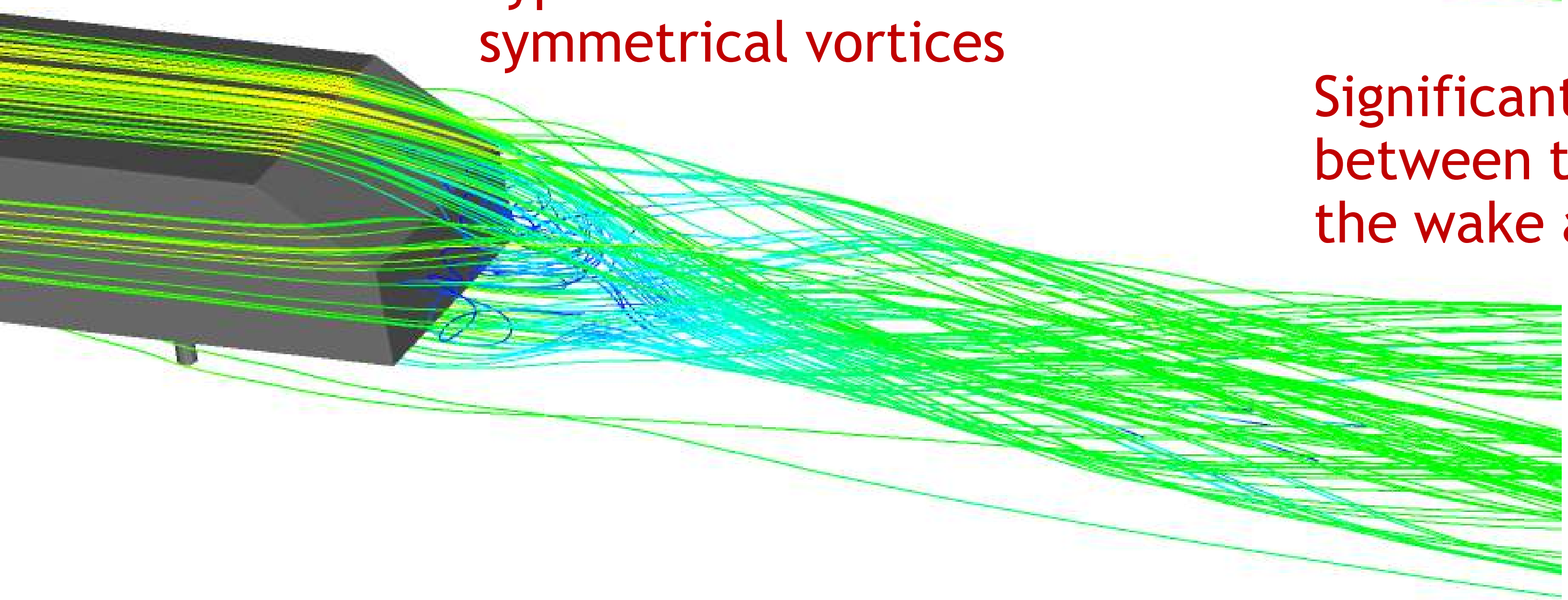
0.00 15.5 31.0 46.5 62.0



25° slant

Typical structures of symmetrical vortices

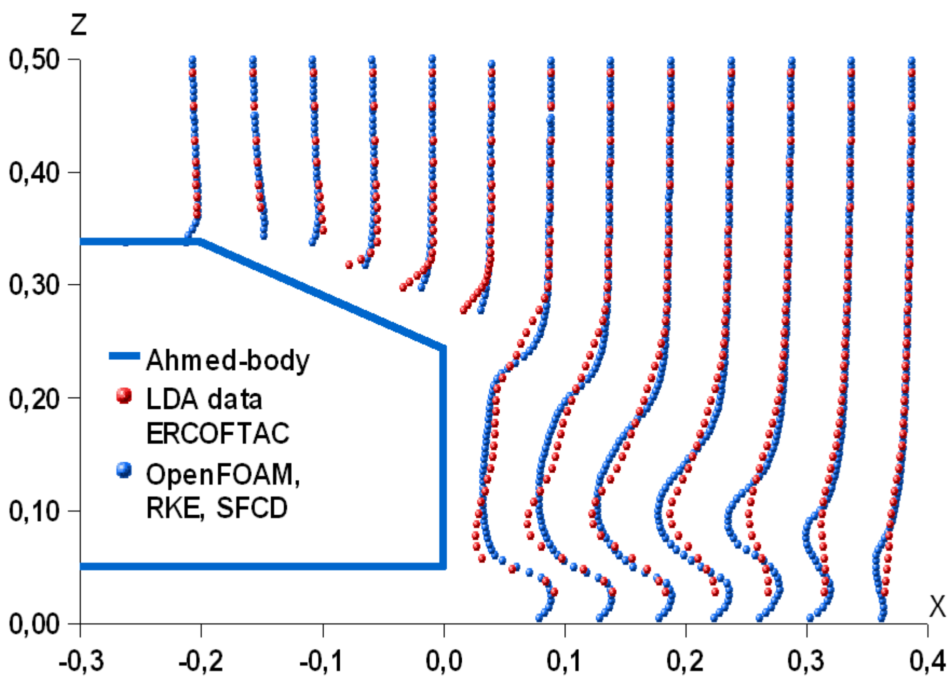
Significant differences between the two cases in the wake area

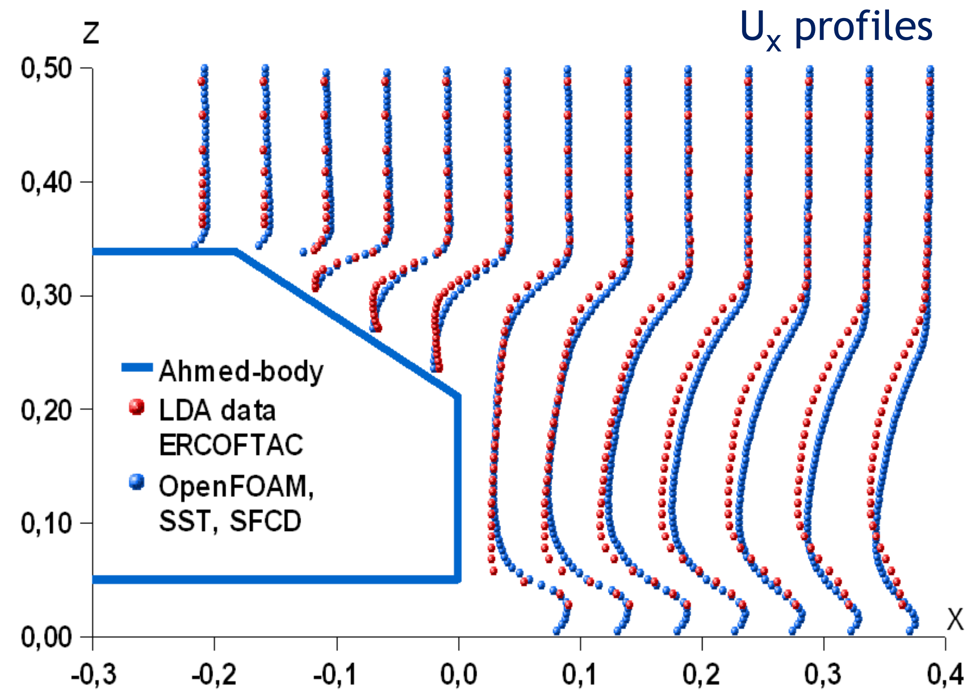


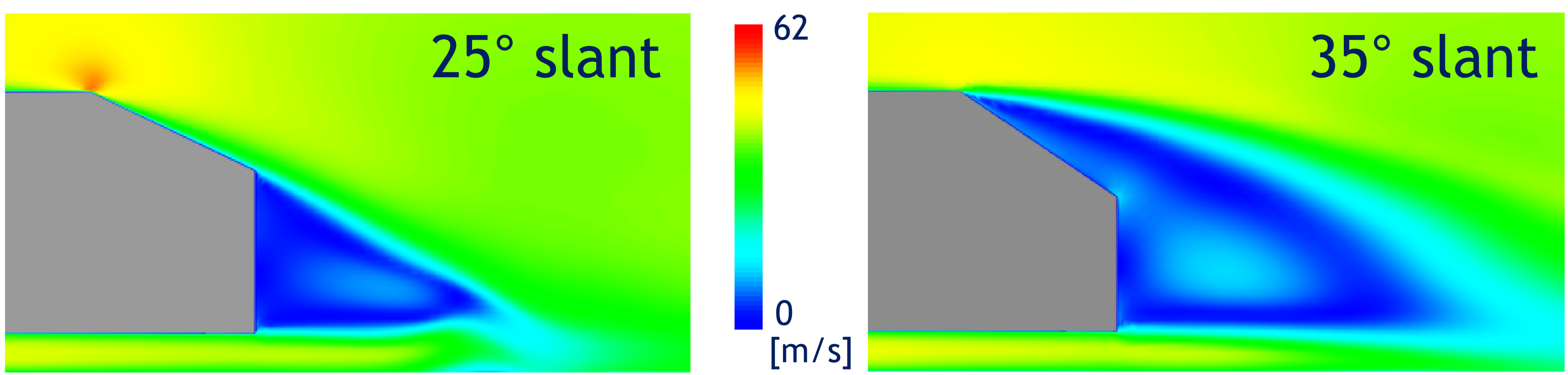
35° slant

Straight flow (2/2)

Velocity fields in symmetry plane





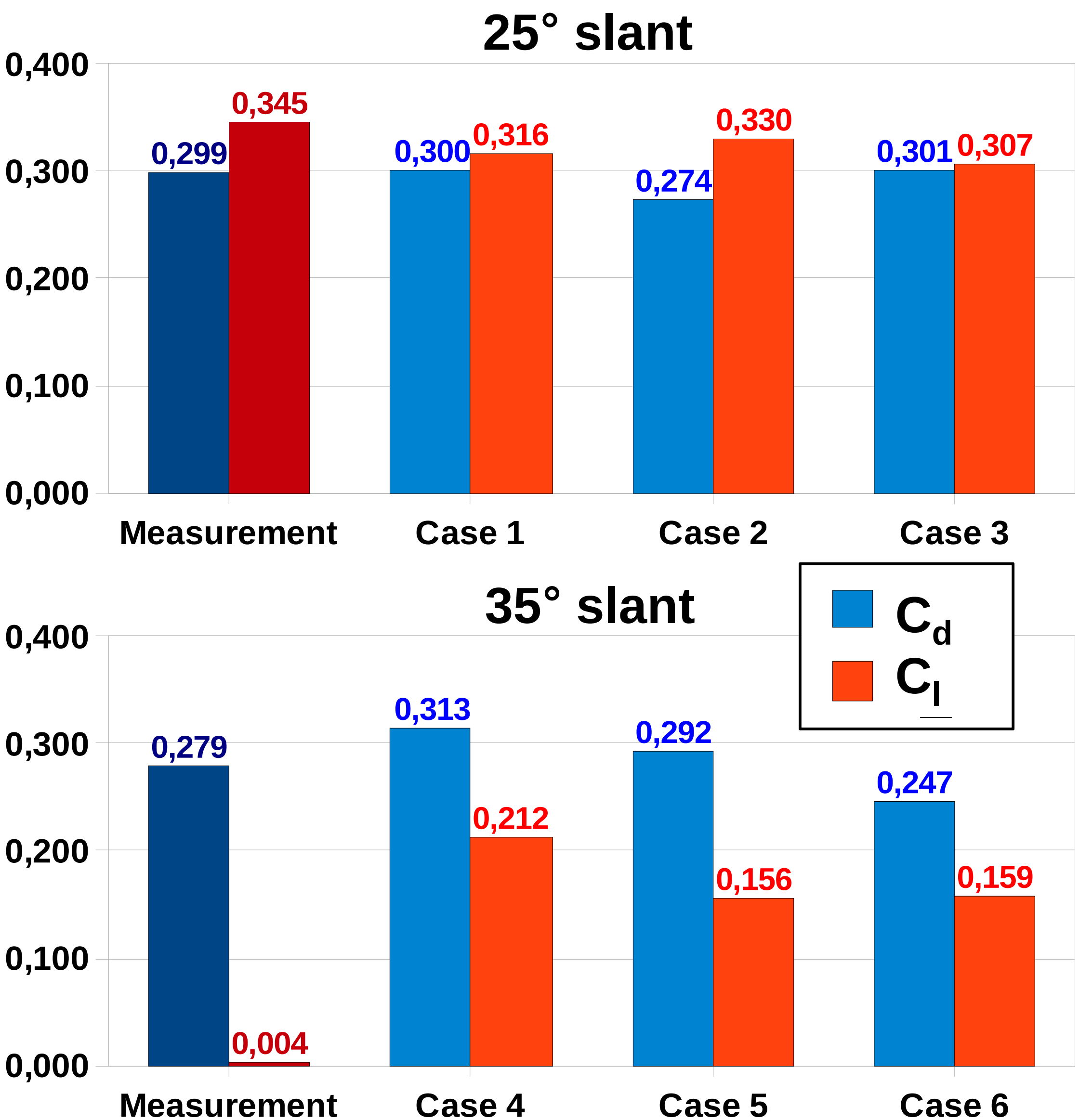


⇨ Attached flow over the slant

⇨ Detached flow over the slant

Mesh variation

Integral drag and lift coefficients



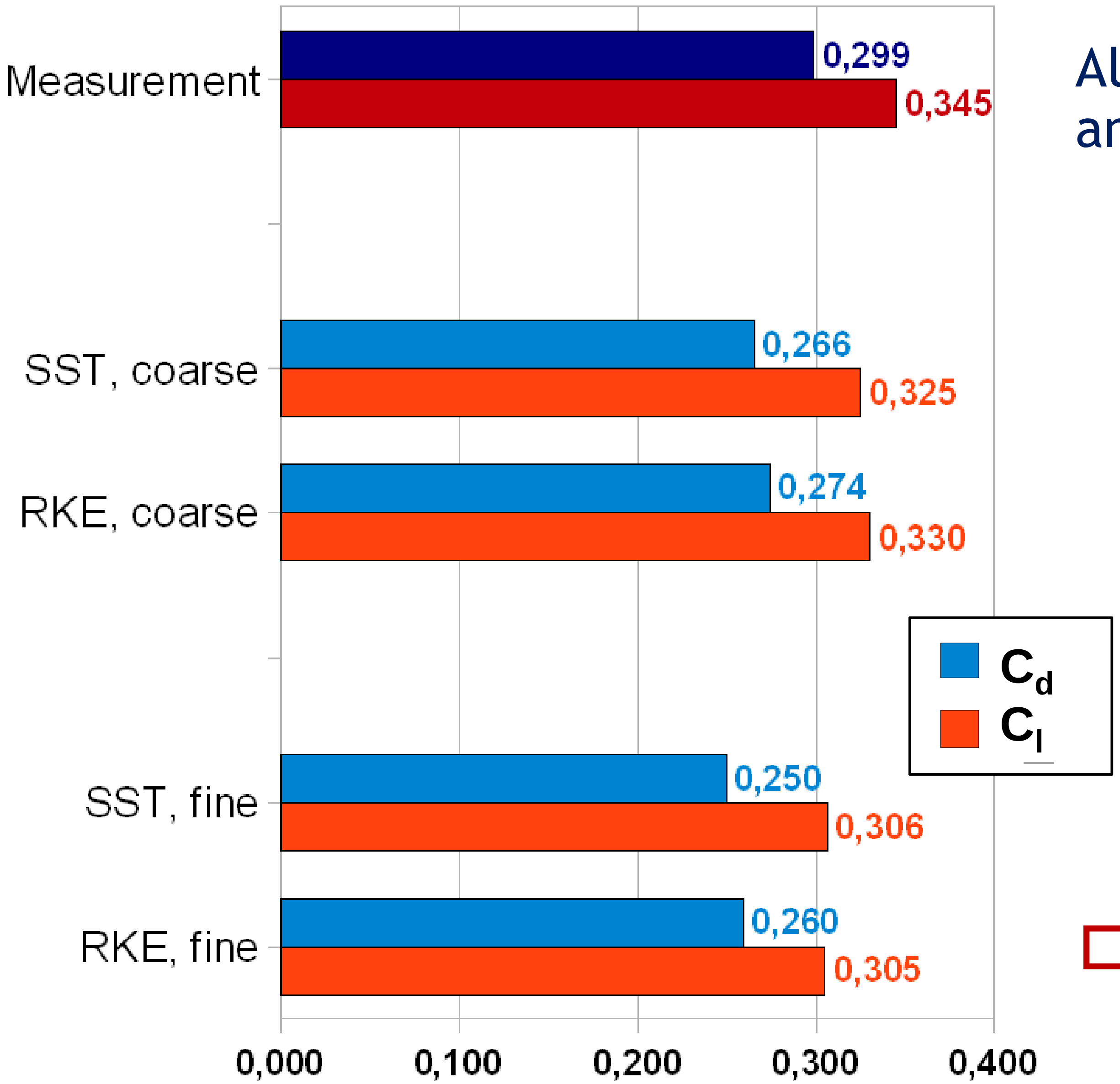
Best results for the following cases:

Ahmed 25°			
Case 1	~1.1 Mio. cells	SST	SFCD
Case 2	~2 Mio. cells	RKE	SFCD
Case 3	~4.7 Mio. cells	SST	upwind
Ahmed 35°			
Case 4	~1.2 Mio. cells	SST	SFCD
Case 5	~2.4 Mio. cells	SST	SFCD
Case 6	~4.6 Mio. cells	SST	limited linear

⇨ Strong deviations from measurements for the lift coefficient in all cases with 35° slant

Turbulence model variation

Integral drag and lift coefficients



All calculations for 25° slant and SFCD Scheme

Turbulence models:

- <u>RKE</u>: realizable-k-ε
- <u>SST</u>: k-ω-SST

Meshes:

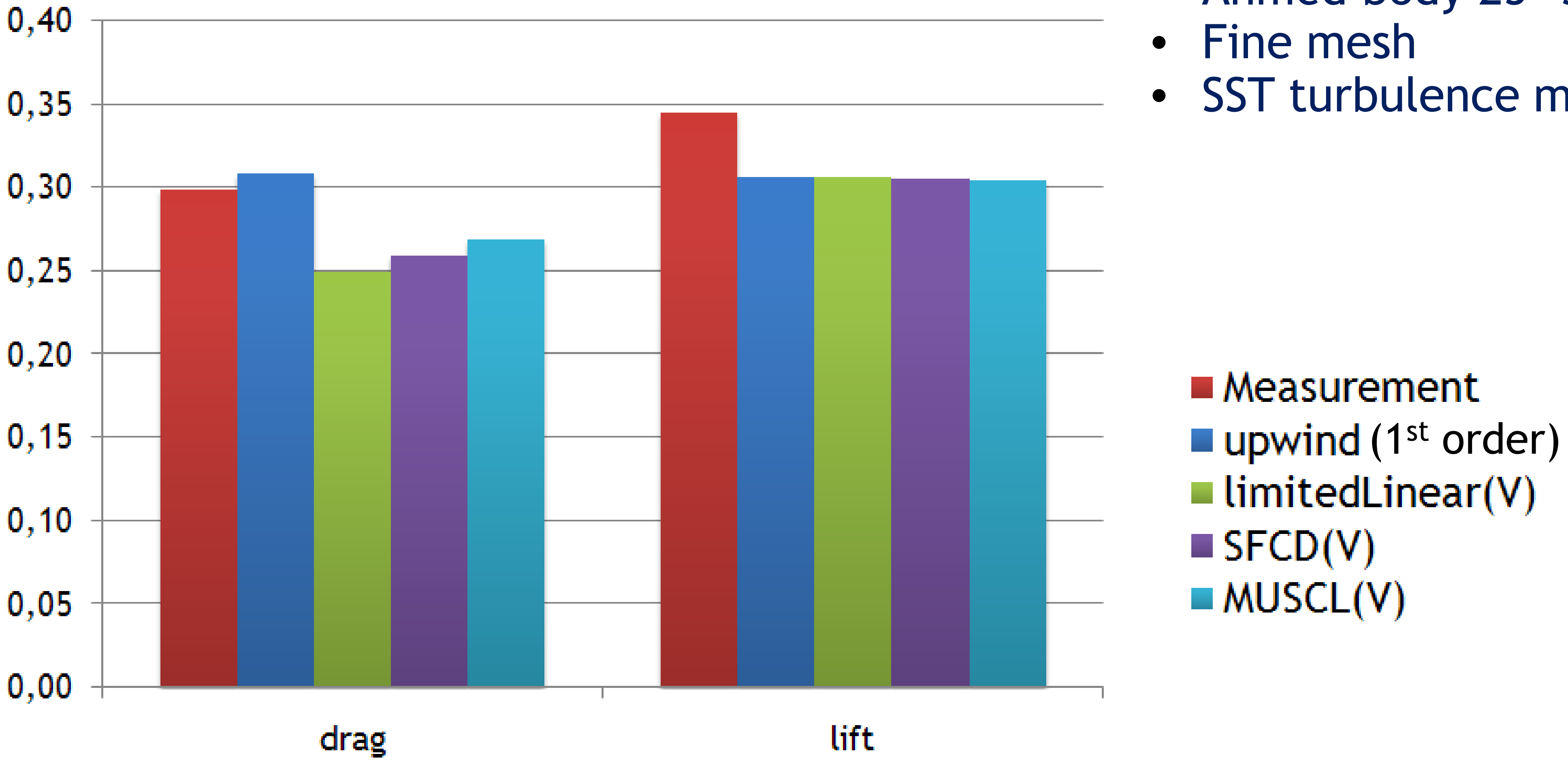
- <u>Coarse</u> (~2 Mio. cells)
- <u>Fine</u> (~4.7 Mio. cells)

⇨ In most cases the RKE model provides slightly higher drag and lift values than the SST model

Convection scheme variation

The Virtual Vehicle Competence Center

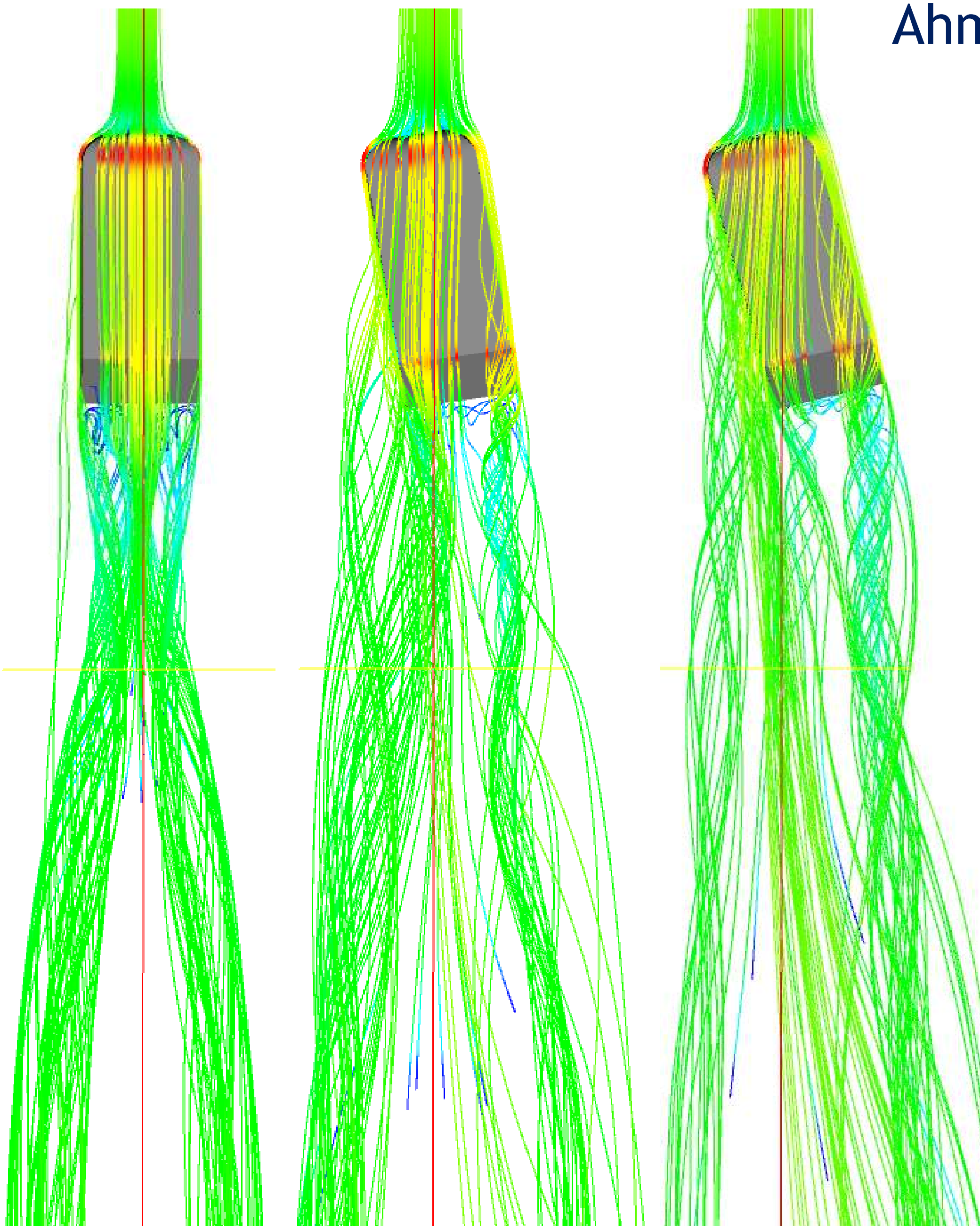
Integral drag and lift coefficients



- Ahmed body 25° slant
- Fine mesh
- SST turbulence model

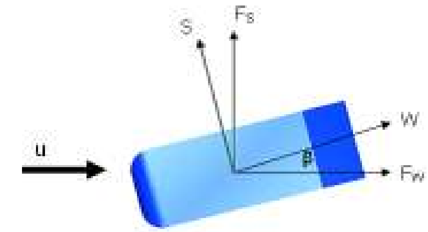
⇨
- The first-order solution provides better results !?
- The MUSCL scheme appears to be the best second order one.

Angle of attack variation

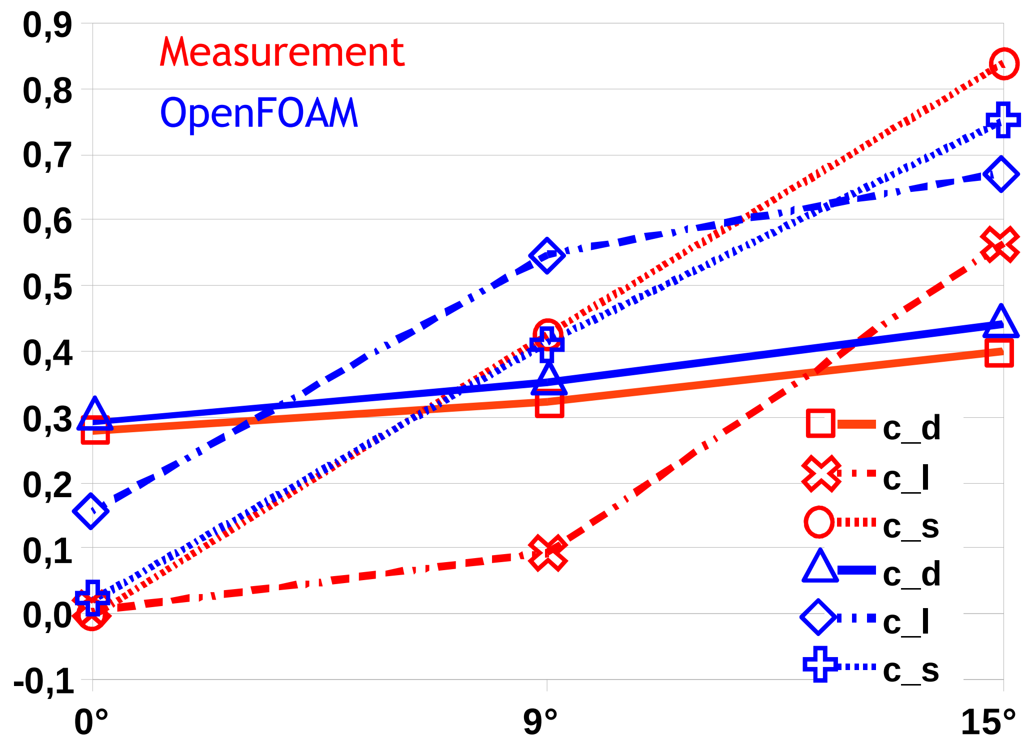


Ahmed body, 35° slant, ~2.5 Mio. cells, SST

	0°	9°	15°
c_d	0,279	0,323	0,400
c_l	0,004	0,093	0,565
c_s	-0,003	0,426	0,838
c_d	0,292	0,354	0,442
c_l	0,156	0,547	0,673
c_s	0,020	0,414	0,751

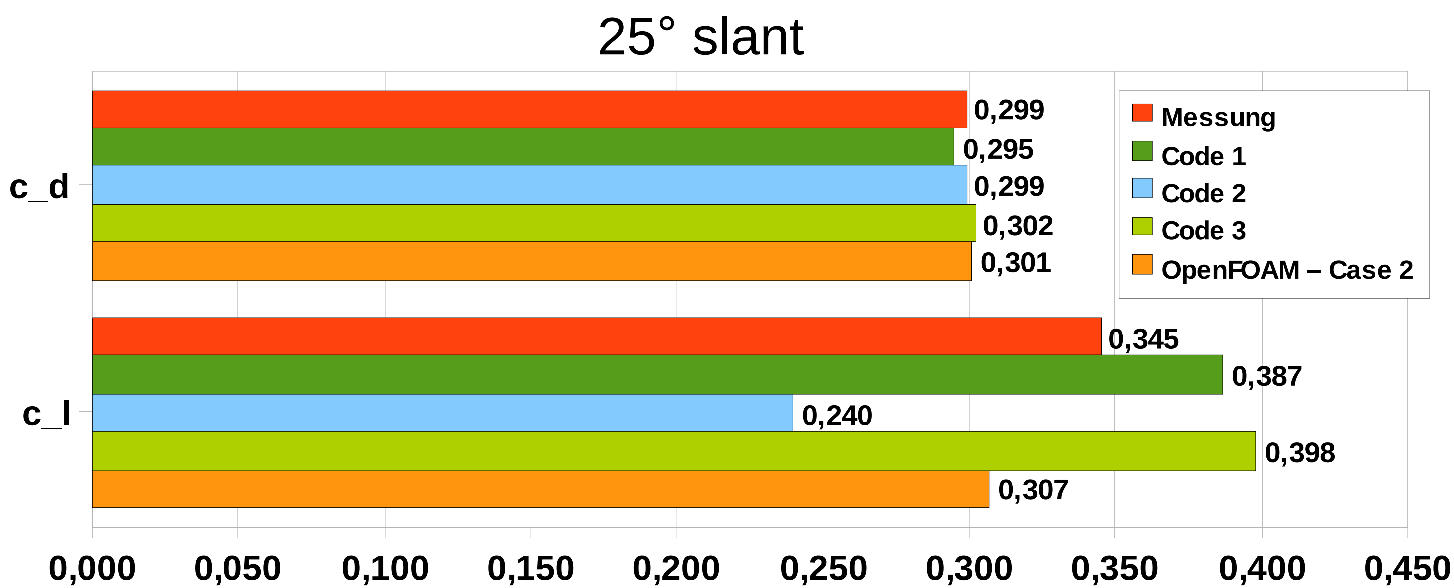


Coefficients referring to Ahmed-body coordinate system



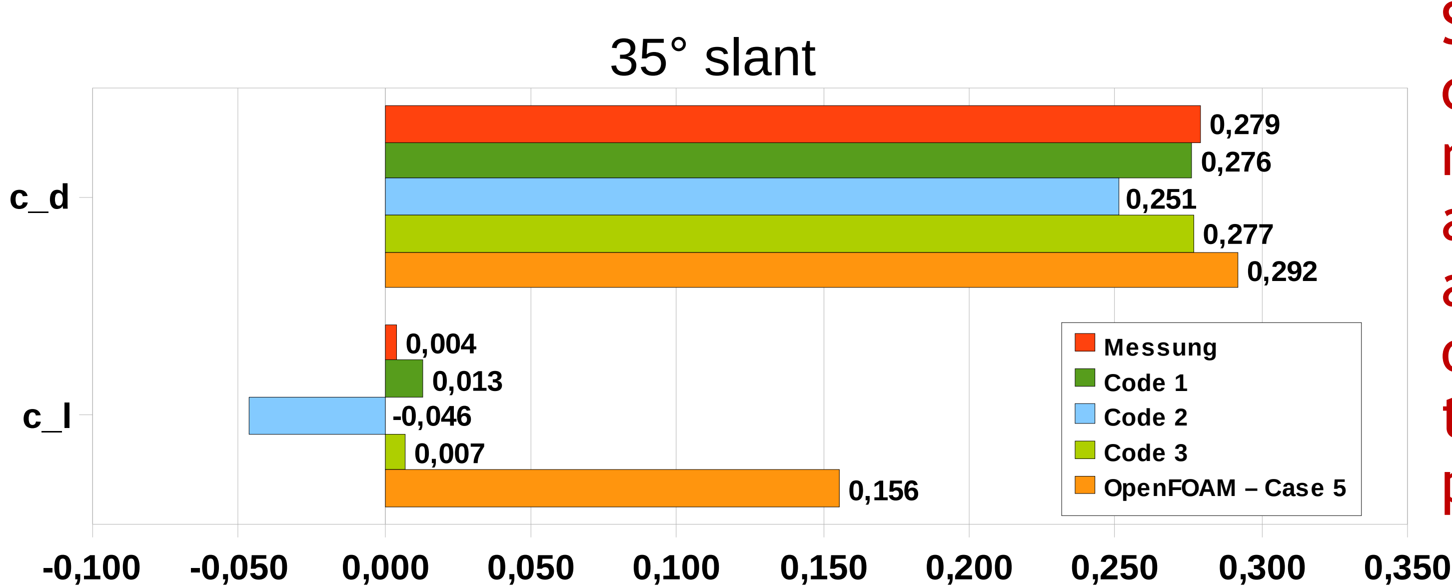
Results from other CFD programs

Integral drag and lift coefficients from different CFD codes



- Internal benchmarks
- steady state

⇩

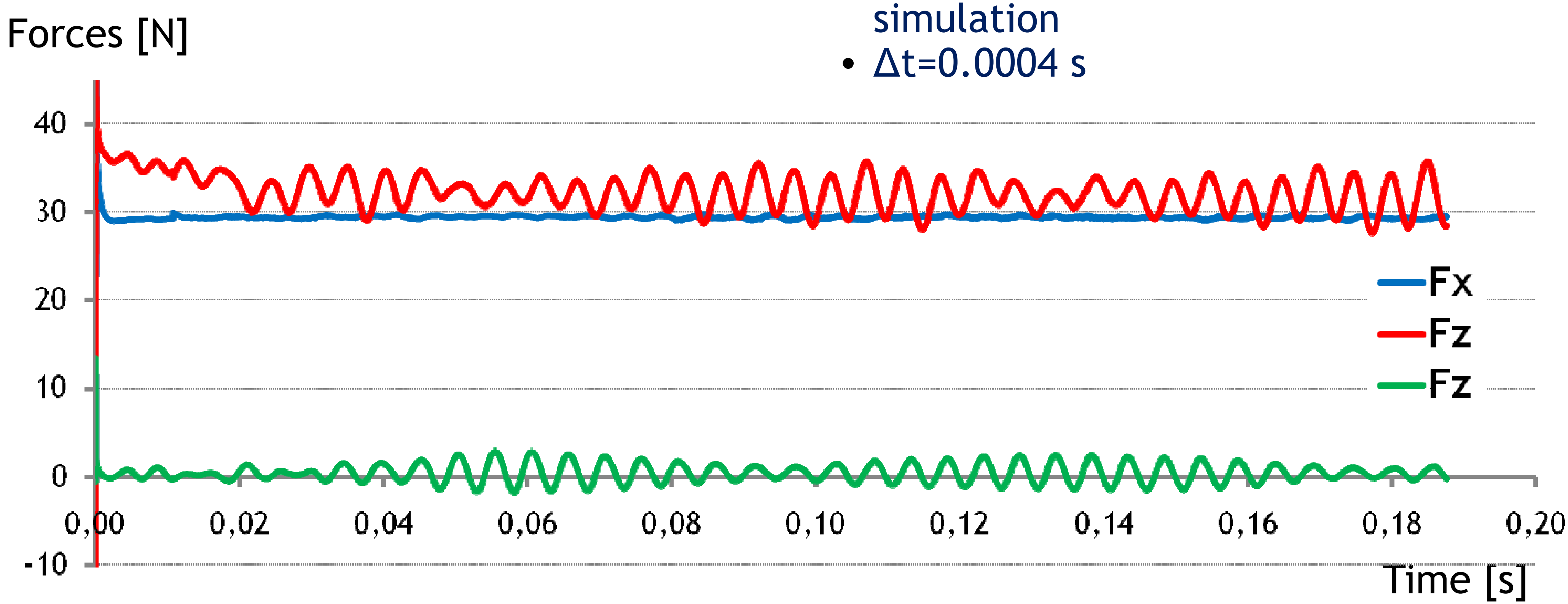


Strong deviation of lift coefficients from measurement data in all cases with 35° slant; apart from this, quality of results is similar to the results from other programs

Unsteady flow simulation (1/2)

Time development of forces on the Ahmed body

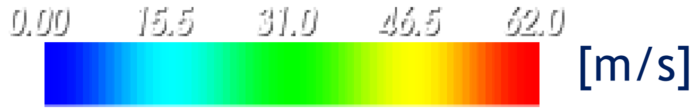
Numerics
- Calculation for 25° slant, 2 Mio. cells, RKE, MUSCL, Crank-Nicolson
- Initialised by the steady state simulation
- $\Delta t = 0.0004$ s



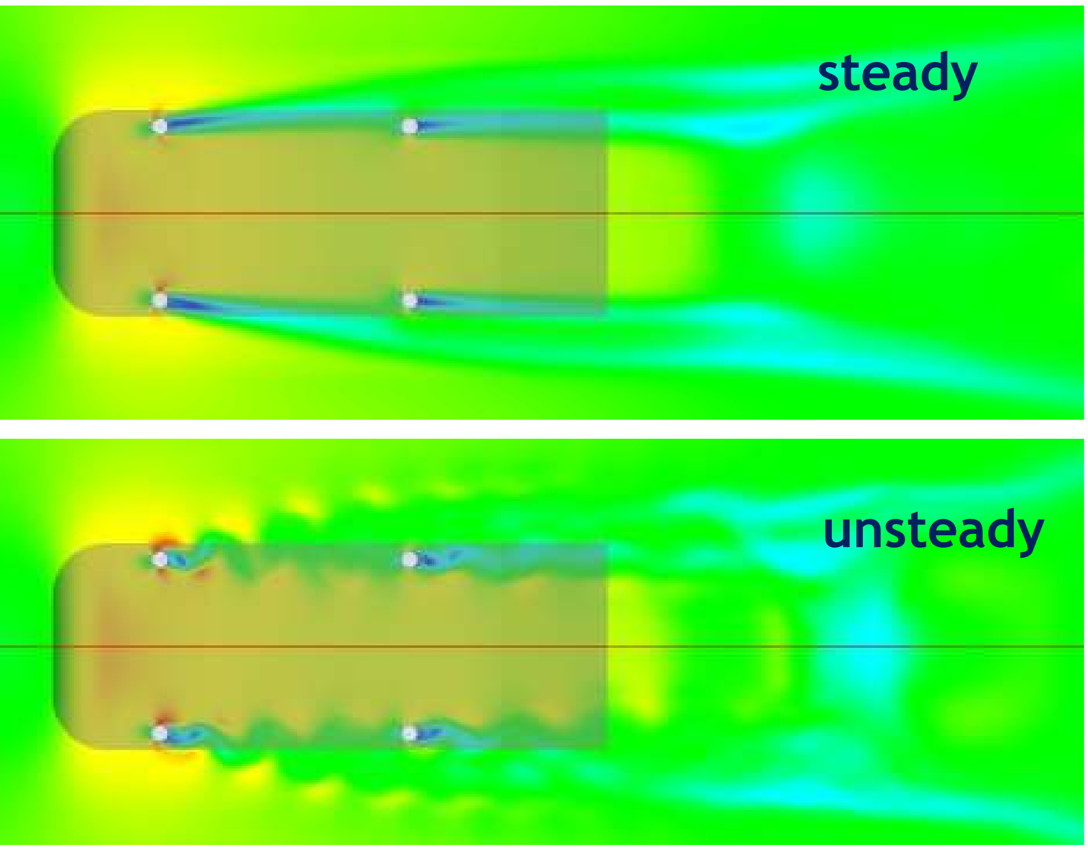
- The simulation reaches a quasi stationary behaviour
- Drag and lift forces are too small compared to the measurements and to the results of steady state simulation with the same mesh

Unsteady flow simulation (2/2)

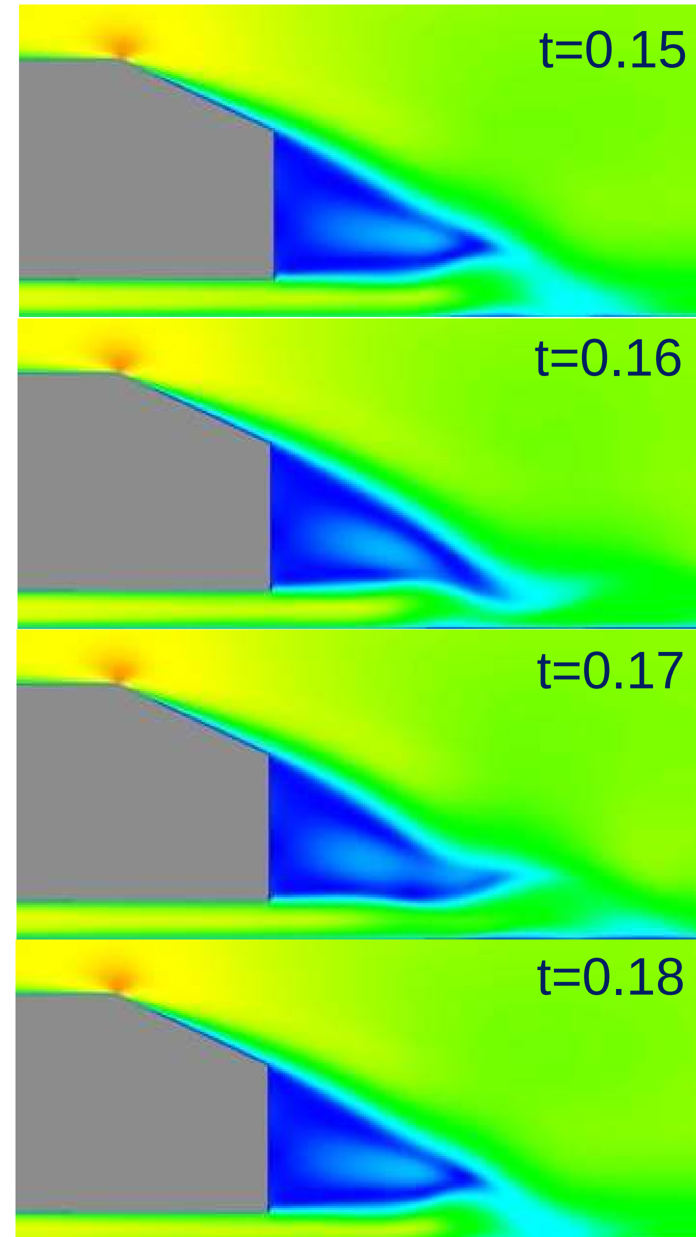
Velocity fields



Plane parallel to the ground



Symmetry plane – wake area



⇨ The local velocity field of the URANS simulation appears more realistic than the one of the steady state result

Overview

Introduction

Test case definition

Numerical method

Results

Conclusions

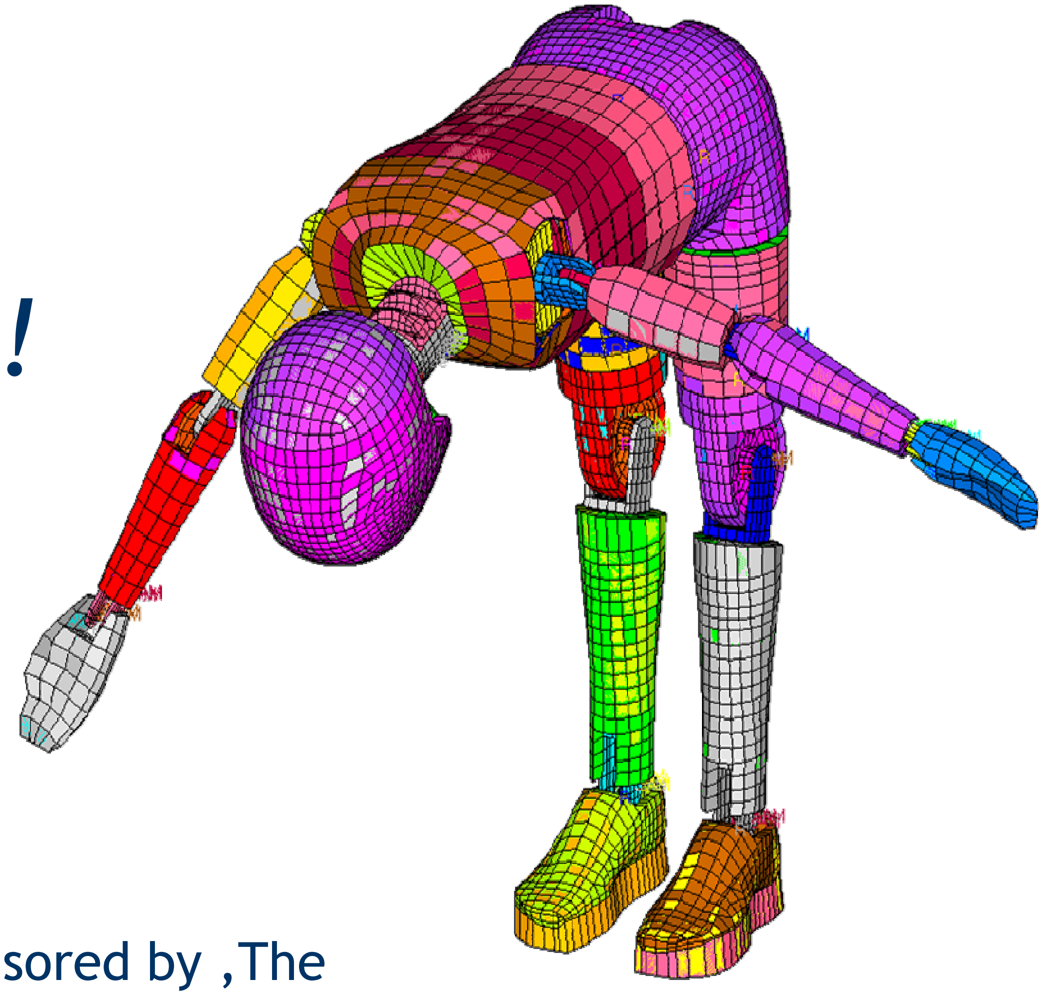
Conclusions

The Virtual Vehicle Competence Center

- **Good simulation results**, except for the lift coefficients in the 35° slant cases
- Full flexibility in modelling and numerics, but
  → good CFD knowledge and user experience are needed;
  → for industrial applications, a unified wall treatment for the whole range of y^+ down to the wall is needed;
  → the graphical interface has still to be enhanced for industrial applications.
- User guides and tutorials are good for the first steps in OpenFOAM, but
  → consistent and complete documentation for the use of components contained in the distribution are needed

⇨ OpenFOAM appears to be a reasonable **supplement** to commercial CFD programs in **industrial applications** with a **high potential of improvement**

Thank you
for your attention!



Questions?

This presentation was created at and sponsored by ‚The Virtual Vehicle Competence Center (vif)‘, Graz/Austria. Initiated by the K plus Competence Center Program and sponsored by Land Steiermark and Steirische Wirtschaftsförderungsgesellschaft mbH.